Autodesk®

RELEASE 2

Autodesk® Civil Design

GETTING STARTED GUIDE

Copyright © 1999 Autodesk, Inc.
All Rights Reserved

This publication, or parts thereof, may not be reproduced in any form, by any method, for any purpose.

AUTODESK, INC. MAKES NO WARRANTY, EITHER EXPRESSED OR IMPLIED, INCLUDING BUT NOT LIMITED TO ANY IMPLIED WARRANTIES OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE, REGARDING THESE MATERIALS AND MAKES SUCH MATERIALS AVAILABLE SOLELY ON AN "AS IS" BASIS.

IN NO EVENT SHALL AUTODESK, INC. BE LIABLE TO ANYONE FOR SPECIAL, COLLATERAL, INCIDENTAL, OR CONSEQUENTIAL DAMAGES IN CONNECTION WITH OR ARISING OUT OF PURCHASE OR USE OF THESE MATERIALS. THE SOLE AND EXCLUSIVE LIABILITY TO AUTODESK, INC. REGARDLESS OF THE FORM OF ACTION, SHALL NOT EXCEED THE PURCHASE PRICE OF THE MATERIALS DESCRIBED HEREIN.

Autodesk, Inc. reserves the right to revise and improve its products as it sees fit. This publication describes the state of this product at the time of its publication, and may not reflect the product at all times in the future.

Autodesk Trademarks

The following are registered trademarks of Autodesk, Inc., in the USA and/or other countries: 3D Plan, 3D Props, 3D Studio, 3D Studio MAX, 3D Studio VIZ, 3DSurfer, ADE, ADI, Advanced Modeling Extension, AEC Authority (logo), AEC-X, AME, Animator Pro, Animator Studio, ATC, AUGI, AutoCAD, AutoCAD Data Extension, AutoCAD Development System, AutoCAD LT, AutoCAD Map, Autodesk, Autodesk Animator, Autodesk (logo), Autodesk MapGuide, Autodesk University, Autodesk View, Autodesk WalkThrough, Autodesk World, AutoLISP, AutoShade, AutoSketch, AutoSolid, AutoSurf, AutoVision, Biped, bringing information down to earth, CAD Overlay, Character Studio, Design Companion, Drafix, Education by Design, Fire, Flame, Flint, Frost, Generic, Generic 3D Drafting, Generic CADD, Generic Software, Geodyssey, Heidi, HOOPS, Hyperwire, Inferno, Inside Track, Kinetix, MaterialSpec, Mechanical Desktop, Mountstone, Multimedia Explorer, NAAUG, Office Series, Opus, PeopleTracker, Physique, Planix, RadioRay, Rastation, Riot, Softdesk, Softdesk (logo), Solution 3000, Stone, Stream, Tech Talk, Texture Universe, The AEC Authority, The Auto Architect, TinkerTech, Vapour, VISION*, *WHIP!, WHIP!* (logo), Wire, Woodbourne, WorkCenter, and World-Creating Toolkit.

The following are trademarks of Autodesk, Inc., in the USA and/or other countries: 3D on the PC, ACAD, ActiveShapes, Actrix, Advanced User Interface, AEC Office, AME Link, Animation Partner, Animation Player, Animation Pro Player, A Studio in Every Computer, ATLAST, Auto-Architect, AutoCAD Architectural Desktop, AutoCAD Architectural Desktop Learning Assistance, AutoCAD Land Development Desktop, AutoCAD Learning Assistance, AutoCAD LT Learning Assistance, AutoCAD Simulator, AutoCAD SQL Extension, AutoCAD SQL Interface, Autodesk Animator Clips, Autodesk Animator Theatre, Autodesk Civil Design, Autodesk Device Interface, Autodesk Inventor, Autodesk PhotoEDIT, Autodesk Software Developer's Kit, Autodesk Survey, Autodesk View DwgX, AutoFlix, AutoPAD, AutoSnap, AutoTrack, Built with ObjectARX (logo), ClearScale, Concept Studio, Content Explorer, cornerStone Toolkit, Dancing Baby (image), Design 2000 (logo), DesignCenter, Design Doctor, Designer's Toolkit, DesignProf, DesignServer, Design Your World, Design Your World (logo), Discreet, DWG Linking, DWG Unplugged, DXF, Extending the Design Team, FLI, FLIC, GDX Driver, Generic 3D, Heads-up Design, Home Series, Kinetix (logo), Lightscape, ObjectARX, ObjectDBX, Ooga-Chaka, Photo Landscape, Photoscape, Plugs and Sockets, PolarSnap, Powered with Autodesk Technology, Powered with Autodesk Technology (logo), ProjectPoint, Pro Landscape, QuickCAD, SchoolBox, SketchTools, Suddenly Everything Clicks, Supportdesk, The Dancing Baby, Transform Ideas Into Reality, Visual LISP, Visual Syllabus, Volo, and Where Design Connects.

Third Party Trademarks

Microsoft, Windows, FoxPro, PowerPoint, Visual Basic, and Windows NT are registered trademarks of Microsoft Corporation.

dBASE and Paradox are trademarks of Borland International, Inc.

Oracle is a trademark of Oracle Corporation.

Lotus 1-2-3 is a trademark of IBM Corporation

All other brand names, product names or trademarks belong to their respective holders.

Third Party Software Program Credits

ACIS ® Copyright © 1994, 1997, 1999 Spatial Technology, Inc. Three-Space Ltd., and Applied Geometry Corp. All rights reserved.

Copyright © 1997 Microsoft Corporation. All rights reserved.

International CorrectSpell ™ Spelling Correction System © 1995 by Lernout & Hauspie Speech Products, N.V. All rights reserved.

InstallShield ™ 3.0 © 1997 InstallShield Software Corporation. All rights reserved.

Portions © 1991-1996 Arthur D. Applegate. All rights reserved.

Portions of this software are based on the work of the Independent JPEG Group.

Typefaces from the Bitstream ® typeface library © 1992.

Typefaces from the Payne Loving Trust © 1996. All rights reserved.

The license management portion of this product is based on Élan License Manager © 1989, 1990, 1998 Élan Computer Group, Inc. All rights reserved.

GOVERNMENT USE

Use, duplication, or disclosure by the U. S. Government is subject to restrictions as set forth in FAR 12.212 (Commercial Computer Software- Restricted Rights) and DFAR 227.7202 (Rights in Technical Data and Computer Software), as applicable.

1 2 3 4 5 6 7 8 9 10

Contents

Chapter 1 Introduction 1

What Is Autodesk Civil Design? 2
Sample Civil Design Projects 2
- Completing a Transportation Engineering Project 3
- Developing a Proposed Grading Plan 4
- Analyzing Existing Surface Water Conditions and Design of Proposed Storm Water Conveyance System 5

Starting Autodesk Civil Design 6
- Menus 7

What's New in Autodesk Civil Design Release 2 8
Finding Information 9
- How to Use Online Help 10
- How to Use the Online Tutorial 14
- Exiting Autodesk Programs 14

Chapter 2 Designing Finished Ground Sites 15

Overview of Grading 16
- Finished Ground Data 16

Creating a Grading Object 17
Editing a Grading Object 20
Creating Contours and Surface Data from a Grading Object 23
Calculating Volumes for a Grading Object 25
Creating a Grading Plan using Daylighting Commands 27
Grading the Surface for a Detention Pond 29
Adding Landscape Symbols to Drawings 31

Chapter 3 Performing Hydrologic Studies 33

Overview of Hydrologic Studies 34
Gathering Data for Hydrologic Analysis 35
Using the Hydrology Calculators 36
Using the Culvert Calculator 38
Using the Rational Method to Calculate Runoff 41
Using the TR-55 Graphical Peak Discharge Method to Calculate Runoff 45
Using the TR-55 Tabular Hydrograph Method to Calculate Runoff 48
Estimating TR-55 Detention Basin Storage 52

Chapter 4 Creating Plan Details 55

Overview of Creating Plan Details 56
Creating Intersections 57
Creating Cul-de-Sacs 59

Chapter 5 Viewing and Editing Roads in Profile View 61

Overview of Viewing and Editing Roads in Profile View 62
Creating Existing Ground Profiles 63
Creating Finished Ground Road Profiles 64
Editing Vertical Alignments 67

Chapter 6 Viewing and Editing Roads in Section View 69

Overview of Viewing and Editing Roads in Section View 70
Creating Existing Ground Sections Along a Road 70
Working with Templates 72
Creating Finished Ground Cross Sections 75
Editing Cross Sections 77
 Edit Design Control 77
 View/Edit Sections 78

Transitioning a Roadway 79
Modifying Roadway Slope 81
Superelevating a Roadway 83
Using Roadway Data for Finished Ground Surfaces 86

Chapter 7 Designing Pipe Runs 89

Overview of Designing Pipe Runs 90
Drawing and Defining Pipe Runs 91
Importing Plan View Pipe Runs 94
Drafting Conceptual Profile Pipe Runs 95
Editing Pipe Runs Graphically 96
Working with the Pipes Run Editor 98
Drafting Finished Plan Pipe Runs 99
Drafting Finished Profile Pipe Runs 101

Chapter 8 Plotting Drawings 103

Overview of Plotting Drawings 104
Working in Model Space and Paper Space 107
Creating Label Styles, Sheet Styles, and Frames 107
- Creating Label Styles 108
- Sheet Styles 108
- Frames 109

Setting Up a Plan/Profile Sheet Style 110
Creating a Plan/Profile Sheet Series 111
Creating a Section Sheet Series 113

Index 115

1

Introduction

Use Autodesk Civil Design with AutoCAD Land Development Desktop to complete site grading plans, hydrologic analysis, and roadway design.



In this chapter

- What is Autodesk Civil Design?
- Sample Civil Design Projects
- Starting Autodesk Civil Design
- What's New in Civil Design Release 2
- Finding Information
- Exiting Autodesk Programs

What Is Autodesk Civil Design?

Autodesk Civil Design Release 2 is part of the Land Development Solutions II suite of products. Autodesk Civil Design Release 2 requires AutoCAD Land Development Desktop Release 2, and uses all the project data created in AutoCAD Land Development Desktop, including points, terrain models, alignments, and so on.

The programs work seamlessly together. You never need to change programs when you need to access a command; you just change menus using the Menu Palette Manager.

Autodesk Civil Design is for people who need advanced civil engineering commands for site grading, hydrological studies, road design, sheet plotting, and pipe design.

Autodesk Civil Design simplifies the creation of:

- Grading plans
- Proposed site plans
- Watershed analysis
- Culvert, weir, and riser design
- Existing ground profile extraction and drafting
- Proposed vertical alignment design
- Roadway sectional design
- Subdivision layout plans
- Proposed roadway plans
- Septic design plans
- Roadway plan, profile, and cross section sheets
- Pipe design plans

This is just a partial list of the plans and reports you can create when you put Autodesk Civil Design to work for you.

Sample Civil Design Projects

You can use Autodesk Civil Design with the AutoCAD Land Development Desktop to complete civil engineering projects as described in the following examples.

Completing a Transportation Engineering Project

Autodesk Civil Design features powerful tools for completing all type and scale transportation (road, rail, runway, channel, and so on) projects. These projects are generally alignment based. For example, a proposed centerline is designed as a base alignment. From that alignment, profiles and sections can be extracted and referenced in the design of the vertical details of the project.

Autodesk Civil Design is fully integrated with AutoCAD Land Development Desktop. This means that the centerline (horizontal) alignment defined in AutoCAD Land Development Desktop can be fully exploited when completing the design process in Autodesk Civil Design. After the alignment is defined in the project, a profile can be extracted and plotted in the drawing. This profile is then used as the basis of your vertical alignment design. There are various options for developing tangents and vertical curves and then defining this vertical alignment in the project.

At this point, you're ready to extract sections along the alignment. After these sections are extracted, you can apply a typical design template and various engineering rules to a range of sections. The results are displayed both graphically and in report form. You can also apply more advanced engineering rules such as superelevation controls, advanced slope controls, and plan or profile transitions for stretching a template to meet plan/profile layout geometry.

To complete the process, plan, profile, and cross sections can be cut from the a combination of design data (project based data) and CAD entities in your drawing.

Developing a Proposed Grading Plan

Autodesk Civil Design offers a broad set of tools to assist you in the design of a proposed grading plan. In some cases, the capabilities work in a way that is very similar to manual methods that you may have used in the past. Other options are highly automated, offering visual and engineering results instantly as you fine tune your design.

Each grading plan will present different challenges. Based on the different design techniques, existing conditions, and site limitations, you can define a proposed grade using grading objects, design points, contours, 3D polylines, and daylighting. In most cases, a combination of these will be the most efficient way of completing your project.

There are various commands you can use from AutoCAD Land Development Desktop menus to set points at a grade or slope, along an entity, or based on interpolation between known elevations. Other commands assist in altering pre-existing point elevations to match a desired grade or slope. Contours can also be used in the development of a proposed grading plan by using copy, offset, and editing functions. In addition, you can use 3D polylines, which are single entities with vertex elevations that can vary. These entities can then be used with other design points, contours, or other 3D entities to build a terrain model.

After a proposed surface is created, contours and other drafting can be completed using the tools provided in either AutoCAD Land Development Desktop or Autodesk Civil Design.

Analyzing Existing Surface Water Conditions and Design of Proposed Storm Water Conveyance System

Autodesk Civil Design provides capabilities to analyze existing surface water conditions across a site, and then layout and analyze a proposed storm water collection system (pipes, structures, ponds). Key to this solution is the integration with the terrain modeling and the graphical layout and editing capabilities of AutoCAD Land Development Desktop.

Various runoff analysis methods are included to meet your regional or project needs. Data such as slope or elevations can be retrieved from the terrain model, and areas and distances can be retrieved directly from entities or graphical selections. Libraries are included to apply other factors such as soil types and land use variables. After all of your data is input, you can generate reports and charts for plotting. The results can also be used in the design of a storm water collection system.

Pipes are laid out graphically across a site or along a road. Each vertex is automatically defined as a structure (manhole, catch basin, and so on) and pipe lengths and rim elevations are automatically extracted. With the pipe run defined, the analysis process can begin and tabular or graphical editing can be performed. Once complete, finished drafting can be created for plan, profile, and cross section plots of the pipe run.

To complete the process, retention and detention ponds can be designed and shaped using a variety of design options. The pond can then be turned into a surface, and the stage-storage results can be integrated into the overall storm water system.

Starting Autodesk Civil Design

Autodesk Civil Design runs within the AutoCAD Land Development Desktop. When you install Autodesk Civil Design, all of your AutoCAD Land Development Desktop commands continue to function as they did before.

To start Autodesk Civil Design after installing it, select the AutoCAD Land Development Desktop R2 icon in the AutoCAD Land Development Desktop R2 program group. All Autodesk Civil Design menus and commands are available for you to use when you load the Autodesk Civil Design menu palette.

To start Autodesk Civil Design

Steps	*Use* Find *to look up*
1 Select the AutoCAD Land Development Desktop R2 icon from the AutoCAD Land Development Desktop R2 program group, or select the AutoCAD Land Development Desktop R2 icon from your Windows desktop. Autodesk Civil Design is combined with the AutoCAD Land Development Desktop when you install Autodesk Civil Design.	
2 From the Projects menu, choose Menu Palettes.	*Select a Menu Palette*
3 Select the Civil Design R2 palette.	
4 Click Load.	
5 Click OK.	

Menus

Autodesk Civil Design adds the following menus to AutoCAD Land Development Desktop:

Pull-down menus included in Autodesk Civil Design

Menu	Functionality
Grading	Perform site grading using grading objects, points, and daylighting; create grading plans for detention ponds
Layout	Create intersections, cul-de-sacs, parking stalls, and sports fields
Profiles	Create existing ground and finished ground profiles
Cross Sections	Create existing ground and finished ground sections
Hydrology	Perform hydrologic site studies using runoff, pipe, channel, culvert, weir, and orifice calculators
Pipes	Create pipes and pipe nodes
Sheet Manager	Set up plan, profile, and section sheets for plotting

You can use the Menu Palettes command on the Projects menu to save a pre-configured group of menus. Use the MENULOAD command to change the location and display of pull-down menus so that they meet your needs. You can then use the Menu Palette Manager to save the changes as a custom menu palette. This palette can then be recalled at any time so that you can restore the menus that are necessary for your project or current task.

For more information about starting new drawings and projects, see the *AutoCAD Land Development Desktop Getting Started Guide*.

What's New in Autodesk Civil Design Release 2

The following topics describe the new features in Release 2 of Autodesk Civil Design.

Changes to Slope Grading

- Graphically add/edit/delete grading object vertices, slope tags, and target regions
- Visual display of current vertex, slope tag, and target region while editing grading properties
- Ability to set footprint elevations based on a fixed elevation or from the average/actual elevations of a surface model
- Streamlined editing options that are available via shortcut menu
- Create contours directly from a grading object
- .dbx foundation enables easy drawing sharing with other applications like Architectural Desktop and 3D Studio VIZ 3

Changes to Sheet Manager

- Changes to take advantage of AutoCAD 2000 Multiple Layouts
- Minor menu modifications to remove unneeded commands such as "Mspace" and "Pspace"

Changes to Cross Sections

- Design Control: Now has left and right bench control
- Design Control: Use of match slopes for left and right side of template
- Superelevation: New documentation outlining the process of calculating superelevation for compound spiral and reverse curve situations

Changes to Hydrology

- Menu reorganization
- SCS Method renamed TR-20 Method
- Defect fixes for use of metric units
- Improved documentation

Changes to Pipes

- Run Editor Settings now includes option to turn off Automatic Pipe Resizing
- Haestad Data Transfer: Haestad SewerCAD and StormCAD products can read and save in the Civil Design pipes.mdb format

Finding Information

The documentation set for Autodesk Civil Design R2 is a combination of online Help files and printed documentation. The following documents are included in your Autodesk Civil Design documentation set.

- *Autodesk Civil Design Getting Started Guide*
- *Autodesk Civil Design User's Guide* (printed and online)
- *Autodesk Civil Design Tutorial* (online)

This guide—the Getting Started Guide—introduces you to Autodesk Civil Design. Each chapter describes a Autodesk Civil Design menu; each section explains how you can use one or more commands to complete a project task. This is not a comprehensive reference manual, but it shows you how you can use Autodesk Civil Design along with AutoCAD Land Development Desktop to complete your civil engineering projects.

Use this guide with the online Help, online tutorials, and courseware to learn how to use Autodesk Civil Design.

Many sections of this guide refer you to topics in the online Help files for more information. For example:



For more information about cross sections, use Find to look up "Overview of Cross Sections" in the online Help.

The  icon indicates that you can find more information using a Help file. The Find icon represents the Find tab on a Help Topics page. Use the search mechanism on the Find tab to locate specific topic titles or topics that match certain keywords.

Some sections in this guide have numbered steps you can perform to complete a task, such as creating a grading plan. The relevant help topic is listed to the right of each step. For example:

To create a grading object

Steps	*Use* Find *to look up*
1 From the Grading menu, choose Slope Grading ➤ Grading Wizard.	*Create a Grading Object using the Grading Wizard*

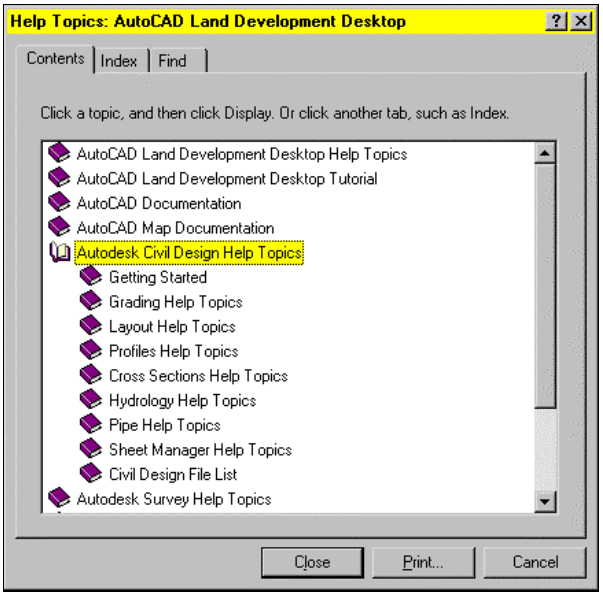
How to Use Online Help

You can access help files for Autodesk Civil Design by using the following methods. The Autodesk Civil Design help files are automatically integrated into the AutoCAD Land Development Desktop interface when you install the program.

Accessing Help Files

Method	Result	Benefits
From the AutoCAD Land Development Desktop R2 program group, select the AutoCAD Land Development Desktop R2 Online Help icon.	Displays a Contents tab that lists the Help files and the tutorial for Autodesk Civil Design, AutoCAD Land Development Desktop, AutoCAD Map, and AutoCAD.	This Help file displays a combined index and table of contents, as well as a combined search mechanism so you can find the Help topics you need.
From within AutoCAD, select Help ➤ Help Topics, type Help at the command line, or press F1.	Displays the same Contents tab as described above.	This Help file has the same benefits as described above.
From a dialog box, click a Help button.	Displays the Help topic that describes how to use the dialog box.	This topic provides the information that you need without having to search for it.
Move your pointer over a command in a menu using the up and down keyboard arrows and press F1.	Displays the Help topic that describes the commands in the menu.	This topic has links to specific Help topics for the commands in the menu.

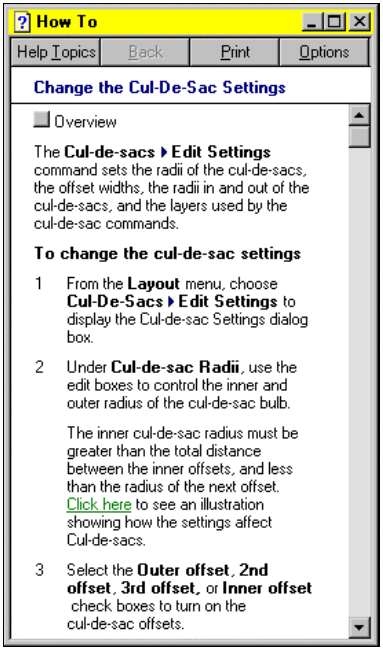
When you open a Help file from either the Help menu or the Help icon, the Help Topics window is displayed, as shown in the following illustration.



This window has three tabs: Contents, Index, and Find.

- Click the Contents tab to view the Table of Contents. This tab has books with topic pages listed underneath each book. To view a topic, double-click the page, or select the page and click Display. You can select a book and click Print to print the all the pages in that book if you would rather have a paper copy of the information.
- Click the Index tab to view an index of Help topics. You can double-click any index entry to view the topic for that entry. If more than one topic shares the same index entry, then you can choose the topic that you want to view.
- Click the Find tab to perform a search on specific words, for example, to search for Help topic titles that are listed in this guide.

The following illustration shows a typical help topic.



When you view a Help topic, you can use the menus and buttons to control options and to navigate. You can also access a shortcut menu by right-clicking in the Help window.

Key Concepts

- When a topic is open, you can move to other relevant topics or definitions by selecting the green, underlined text.
- You can click Back to move to the previous topic that you viewed.
- You can click Help Topics to return to the Help Topics window.
- To print a topic that is displayed in a popup window (a Help window that has no menu options and disappears if you click elsewhere on your screen), right-click to display a shortcut menu and click Print.

The following task shows you how to locate a topic title in the Help file.

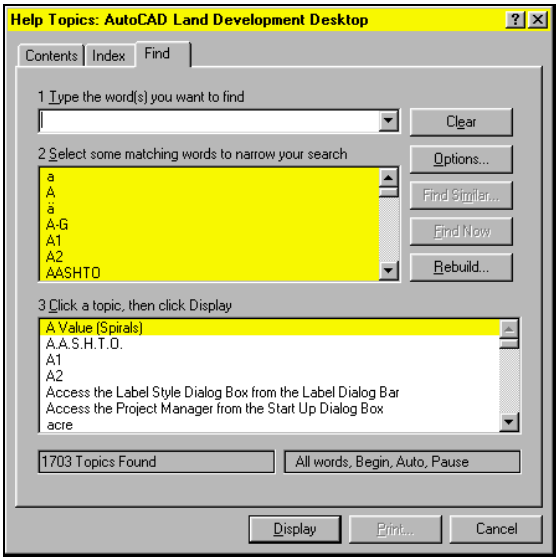
To use the online Help to locate a topic title

Steps

1 Start Help by using one of the methods listed in the Accessing Help Files list in "How to Use Online Help" in this chapter.

2 Click Find.

The Help window appears as shown in the following illustration.



NOTE If you have not previously used the Find tab, then the Find Setup Wizard prompts you to create a word search database. Click Next to proceed through the wizard. When the wizard has created the database, the Find tab is displayed.

3 In the first box on the Find tab, type the Help topic title that you want to find. Each topic that has similar keywords is displayed in the third box on the dialog box.

To use the online Help to locate a topic title (*continued*)

Steps

4 Click the name of the topic that you want to read, and then click Display to view the Help topic.

5 You can print the topic by selecting Print; you can view relevant topics by clicking the green, underlined text; or you can return to the Find tab by clicking Help Topics.

How to Use the Online Tutorial

AutoCAD Civil Design R2 has an online tutorial that you can use to learn the program's concepts. The online tutorial is an excellent way to become familiar with the program.

Open the online Help by using one of the following methods:

- From the AutoCAD Land Development Desktop R2 program group, select the AutoCAD Land Development Desktop R2 Online Help icon.
- From within the program, select Help ➤ Help Topics.
- Press F1 when no command is running or highlighted on the menu.

When the main Help Contents tab is displayed, open up the Autodesk Civil Design Tutorial book, and then open a lesson.

The tutorial is displayed in a small window that you can keep open on top of your screen while you perform the steps. If you want to hide the window while you perform the steps, then select Options ➤ Keep Help On Top ➤ Not on Top.

Click  to move to the next task.

Exiting Autodesk Programs

You can exit Autodesk Civil Design and AutoCAD Land Development Desktop by using any of the following methods:

- From the File menu, choose Exit.
- Type **exit** or **quit** at the command prompt.
- Click the close box in the upper-right corner of the AutoCAD Land Development Desktop window.

2

Designing Finished Ground Sites

Use the commands on the Grading menu to create grading objects, to calculate daylighting information, and to create and shape detention pond definitions.



In this chapter

- Grading Overview
- Creating a Grading Object
- Editing a Grading Object
- Creating Contours and Surface Data from a Grading Object
- Calculating Volumes for a Grading Object
- Creating a Grading Plan using Daylighting Commands
- Grading the Surface for a Detention Pond
- Adding Landscape Symbols to Drawings

Overview of Grading

Developing a grading plan typically results in the creation of a proposed surface model. This allows you to analyze a site efficiently and accurately and to create reports, graphics, and 3D presentation materials that are necessary for the completion of the project. Using this finished ground model, you can calculate cut and fill volumes, determine grading limits, generate proposed grade and cut/fill contours, calculate the watershed areas for the surface, and create post-development runoff models.

Finished Ground Data

Whereas an existing ground surface is based on surveyed points and existing contours, a finished ground surface is based on grading data that you create. Your goal is to create enough grading data so that this finished ground surface is as accurate as possible. Grading data can consist of points, 3D polylines, contours, pond models, daylight lines, points, and breaklines.

There are many commands in AutoCAD Land Development Desktop that you can use to create grading data, including points, contours, and 3D polylines.

Autodesk Civil Design adds the ability to create the following grading data:

- Grading objects
- Daylight lines, points, and breaklines
- Finished ground labels
- Pond models that you can use for hydrology calculations

When you have created all of the finished ground grading data, you can then create the finished ground surface. For more information about creating surfaces, see Chapter 5, "Working with Surfaces" in the *AutoCAD Land Development Desktop Getting Started Guide*.

Creating a Grading Object

Grading objects are three-dimensional objects that represent finished ground grading schemes. You can create a grading object by drawing a footprint, defining slopes, and defining the grading targets (which are the elevations, distances, or a surface that you want to grade to) to generate the 3D information. After you've generated a grading object, you can create contours, breaklines, and surfaces from the 3D information.

The first step in creating a grading object is drawing a footprint. The footprint represents the outline of the object you want to grade from. It can be a 2D or 3D polyline, line, or arc (you can also grade from the daylight of an existing grading object).

The footprint stores elevational information at the vertices and interpolates elevations along the segments between the vertices. During the design process you can edit the vertex elevations. When you use a 2D polyline with embedded arc segments as a grading footprint, the geometry of the arcs is stored within the grading object. The elevations of the arc endpoints can be changed (to represent curbs or fillets in 3D), while still maintaining the true 2D geometry of the original arc.

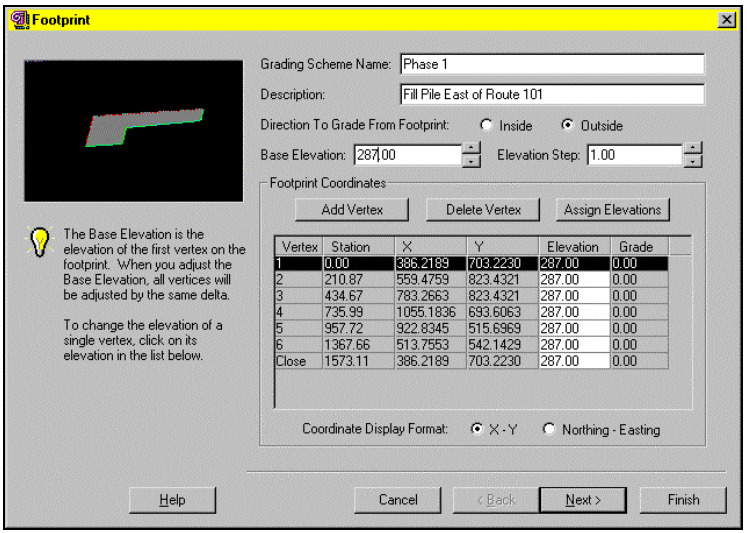
After drawing the footprint, you can run the Grading Wizard. Using the wizard, you can define footprint elevations and then you can select the target you want to grade to. Using target regions you have the option to grade to multiple targets, such as a surface, an elevation, and a distance. Using slope tags, you can create slopes that smoothly transition from one grade to another.

Key Concepts

- Grading objects can be created from open or closed footprints.
- You can create a grading object using one of two methods. The Grading Wizard steps you through every setting you need to establish, and then creates the grading object. Or you can use the two step process of changing the settings and then applying grading.
- After you create a grading object you can make changes in the grading properties or use grips to graphically make edits.
- From a grading object, you can create surfaces and breaklines.
- You can calculate general volume statistics for a grading object if its grading target is a terrain surface or an absolute elevation.

To create a grading object using the Grading Wizard

Steps	*Use* Find *to look up*
1 From the Grading menu, choose Slope Grading ➤ Grading Wizard. Use the Next and Back buttons to move through the sheets.	*Create a Grading Object using the Grading Wizard*



2 On the Footprint sheet, enter a Grading Scheme Name and Description for the footprint. Select Inside or Outside (or Right or Left if the footprint is open) for the direction you want to grade from the footprint. Change the Base Elevation of the footprint and edit vertex elevations, if necessary.	*Configure the Grading Footprint Settings*
3 On the Targets sheet, select the target you want to grade to, a surface, an elevation, or a distance. You can add and delete target regions, if necessary.	*Configure the Grading Targets Settings*
4 On the Slopes sheet, enter the Cut Slope and Fill Slope. You can add and delete slope tags and edit stations.	*Configure the Grading Slopes Settings*

To create a grading object using the Grading Wizard (*continued*)

Steps	*Use* Find *to look up*
5 On the Corners sheet, choose a global corner treatment, or enter corner treatments for individual corners.	*Configure the Grading Corners Settings*
6 On the Accuracy sheet, select a method for spacing, and enter increment values for the projection lines.	*Configure the Grading Accuracy Settings*
7 On the Appearance sheet, select the color, visibility, and linetype for the grading object components and select the grips you want visible in the drawing.	*Configure the Grading Appearance Settings*

NOTE The sheets in the Grading Wizard correspond to the tabs in the Grading Properties and Settings dialog boxes.

To create a grading object using grading settings

Steps	*Use* Find *to look up*
1 From the Grading menu, choose Slope Grading ➤ Settings.	*Create a Grading Object using the Grading Settings*
2 Select the tabs at the top of the dialog box to enter settings for the footprint, targets, slopes, corner treatments, accuracy, and appearance.	*Overview of Configuring the Grading Settings*
3 From the Grading menu, choose Slope Grading ➤ Apply Grading to apply the settings and create a grading object.	*Create a Grading Object using the Grading Settings*



For more information about grading settings, use Find to look up "Overview of Configuring Grading Settings" in the online Help.

Editing a Grading Object

If you want to make changes to the grading object after you have created it, you can change the Grading Properties, or use grips to graphically edit the grading object. Using the grading object shortcut menu is another way you can make changes to the vertices, slope tags, and target regions. To edit a grading object it must be unlocked.

You can also make changes to a grading object using AutoCAD editing commands, including grip editing. If the grading object is unlocked, it will automatically update. If the grading object is locked, you can make changes, but the changes will not be reflected in the drawing until you unlock the grading object.



For more information about locking and unlocking the grading object and using AutoCAD editing commands, use Find to look up "Overview of Grading Object Locking" and "AutoCAD Editing Commands for the Grading Object" in the online Help.

Key Concepts

- You can change the grading properties to edit a grading object, or you can make edits graphically using the grading object grips or shortcut menu commands.
- You can choose which grips you want visible by changing the appearance settings in the Grading Properties.
- Certain grips on a grading object cannot be edited, such as the first and last station for a target region, and the first slope tag location grip.
- Slope tag location grips cannot be moved past a target region grip or past another slope tag location grip. The distance between grips is determined by the Minimum Region Length in the targets settings.

To edit a grading object's properties

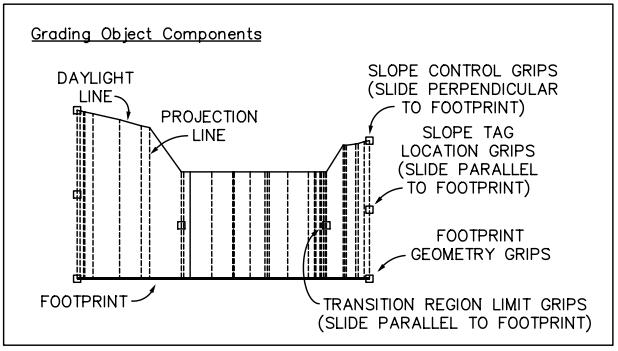
Steps	Use Find to look up
1 Select a grading object in your drawing, then right-click to access the grading object shortcut menu.	
2 Click on Grading Properties.	
3 Modify the properties as needed. When you exit the Grading Properties dialog box, the grading object is updated with the changes.	*Overview of Configuring the Grading Settings*

To grip edit a grading object

Steps	Use Find to look up
1 Select a grading object in your drawing.	*Overview of Using Grips to Edit Grading Objects*

2 Select the grip you want to edit.

The following illustration shows the location of grading object grips.



TIP You can choose which grips are displayed on a grading object by changing the appearance settings in the Grading Properties.

3 Move the grip to edit the grading object. The next time you display the Grading Properties, notice that the spreadsheet sections reflect the changes you made using grips.

To edit a grading object using the shortcut menu

Steps	*Use* Find *to look up*
1 Select a grading object in your drawing.	*Overview of Editing a Grading Object using the Shortcut Menu*

2 Right-click to display the grading object shortcut menu.



For more information about editing grading objects, use Find to look up "Overview of Editing Grading Objects" in the online Help.

Creating Contours and Surface Data from a Grading Object

If you want to use the grading object's 3D information in a terrain model surface, you have several options. You can create a new surface from the grading object, you can create contours, or you can create breakline data from the grading object for any new or existing surface.

Surfaces are created using 3D information from the grading object footprint, daylight lines, and projection lines. The footprint and projection lines are treated as breaklines. The daylight line is treated as a boundary. After you have created the surface it has the same functions as other surfaces, and you can manage the surface from within the Terrain Model Explorer.

Using the Create Contours command, you can directly create contours from a grading object without having to first create a terrain model surface. When you use the Create Contours command a temporary surface is created using the daylight line as the surface boundary. The contours are generated from this temporary surface and then the surface is discarded.

Breaklines can be created from a grading object and added to the current surface, to a new surface, or to any existing surface. When you create breaklines from a grading object, the breakline information is determined from the grading object footprint, daylight lines, and projection lines.

To create a surface from a grading object

Steps	*Use* Find *to look up*
1 Create a grading object. For more information, see "Creating a Grading Object" in this chapter.	
2 From the Grading menu, choose Slope Grading ➤ Create Surface to display the New Surface dialog box.	*Create a Surface from a Grading Object*
3 Type a name and an optional description for the surface and click OK. The surface is created and built.	

To create a surface from a grading object (*continued*)

Steps	*Use* Find *to look up*
4 To view the surface details, use the Terrain Model Explorer. From the Terrain menu, choose Terrain Model Explorer.	
5 In the left pane of the Terrain Model Explorer, open the folder of the surface you created from the grading object to see the surface details.	

To create contours from a grading object

Steps	*Use* Find *to look up*
1 Create a grading object. For more information, see "Creating a Grading Object" in this chapter.	
2 From the Grading menu, choose Slope Grading ➤ Create Contours.	*Create Contours from a Grading Object*
3 In the Create Contours dialog box, enter contour data.	*Overview of Creating Contours from a Surface*

To create breaklines from a grading object

Steps	*Use* Find *to look up*
1 Create a grading object. For more information, see "Creating a Grading Object" in this chapter.	
2 From the Grading menu, choose Slope Grading ➤ Create Breaklines.	*Create Breaklines from a Grading Object*
3 Do one of the following: Type **Current** to add breaklines to the current surface. Select the grading object and enter a description for the breaklines. Type **New** to add the breaklines to a new surface. The New Surface dialog box is displayed. Enter a name and a description for the new surface and click OK. Type **Select** to add the breaklines to an existing surface. The Select Surface dialog box is displayed. Select the surface you want the breaklines to be added to and click OK.	

Calculating Volumes for a Grading Object

You can calculate general volume statistics for the grading object using the Statistics tab in the grading properties or the Calculate Volumes command. The composite volume method is used to calculate the volume results. This method compares the grading object with the grading target(s) to determine the volumes.

The grading object must meet certain requirements in order for the Calculate Volumes command or the Calculate button on the Statistics tab to work properly. In instances where these commands do not generate volumes, or if you want to verify volume calculations, you can create a surface from the grading object (and add surface information to the interior of the footprint, such as points, contours, or 3D polylines if needed), and then use the Volume commands on the Terrain menu to calculate volumes.

Key Concepts

- Volumes are only calculated under the following conditions:
 - If the target is a surface and the grading direction is to the outside of a closed footprint, volumes will be calculated between the object and the surface
 - If the target is an absolute elevation, volumes will be calculated between the object and the elevation
- Volumes are not calculated under the following conditions:
 - If the grading object has multiple targets
 - If the grading object has a single relative elevation target
 - If the footprint is closed and graded to the inside using a surface target
 - If the daylight line(s) cross and the condition is detected by the program
- For more accurate volume calculations, specify smaller line and arc increments on the Accuracy tab of the Grading Properties dialog box.
- Calculate final volumes using the Volume commands on the Terrain menu.



For more information on calculating volumes, use Find to look up "Calculate Volume Data for a Grading Object" in the online Help.

To calculate volumes

Steps	*Use Find to look up*
1 Create a grading object. For more information, see "Creating a Grading Object" in this chapter.	
2 Create a surface from the grading object. For more information, see "Creating Contours and Surface Data from a Grading Object" in this chapter.	
3 From the Grading menu, choose Slope Grading ➤ Calculate Volumes.	
4 From the Grading menu, choose Grading Properties and select the statistics tab. The volume statistics are automatically generated.	

Creating a Grading Plan using Daylighting Commands

As an alternative to using the grading object to create grading plans, you can use the Daylighting commands. The following example explains how to use these commands to draw the outline of a building pad and then project slopes down to match the existing ground.

Key Concepts

- An existing ground surface model is required for using the Daylighting commands.
- You can use either lightweight, 2D, or 3D polylines to draw the footprint outline.
- The program projects perpendicularly from each vertex location on the polyline to the surface model. The more vertices, the better the proposed daylight matchline.

To create grading plans using daylighting commands

Steps	*Use* Find *to look up*
1 From the Grading menu, choose Daylighting ➤ Select Daylight Surface to select into which surface the slopes will match.	*Select the Daylight Surface*
2 Use the 3D polylines commands in the Terrain ➤ 3D Polylines menu to create the proposed design. Draft your proposed outline using 3D polylines either at a continuous elevation, or changing elevations.	*Overview of Creating 3D Polylines*
3 From the Terrain menu, choose 3D Polylines ➤ Fillet 3D Polyline to fillet (round) the corners of the outline if necessary. This will create more daylight projections radially around each corner.	*Fillet 3D Polyline Vertices*
4 From the Grading menu, choose Daylighting ➤ Add Vertices to add more vertices to the polyline outline. The closer the vertices, the more accurate the daylight slopes.	*Add Vertices to a Polyline for Daylighting*

To create grading plans using daylighting commands (*continued*)

Steps	*Use* Find *to look up*
5 From the Grading menu, choose Daylighting ➤ Create Single to determine the daylight matchline at a specified slope. Single applies a constant slope to the entire polyline footprint. The command automatically checks for both cut and fill. As the command runs, temporary objects are drawn that represent the location where the projected slope matches into existing ground.	*Calculate Daylight Points Based on a Single Slope*
6 From the Grading menu, choose Daylighting ➤ Create Multiple if you need to daylight using different slopes. For instance, if one area of the proposed plan falls outside of your construction limits (i.e. property line or building), you can change an individual slope or group of projected slopes. Temporary objects are drawn that show the new daylight matchline location.	*Calculate Daylight Points Based on Multiple Slopes*
7 To insert objects into the drawing that represent the grading plans, you can use the Daylight All command to import a 3D daylight matchline and proposed grading points and breaklines. You can then use these objects to create the proposed ground surface model.	*Insert Daylight Points, Breaklines, and Polylines into a Drawing*

Grading the Surface for a Detention Pond

You can use the detention pond design features of Autodesk Civil Design to design retention and/or detention ponds for controlling peak flow rate amounts from watersheds.

Before you begin the grading plan for the detention pond, determine the design criteria for the pond, such as the volume of water that the pond has to store. To do this you can use the Autodesk Civil Design Hydrology commands. You can use the Graphical Peak Discharge, Rational, or Tabular Hydrograph Methods, or you can determine this information from inflow hydrograph and outflow hydrographs. For more information about calculating pond storage volume, see Chapter 3, "Performing Hydrologic Studies."

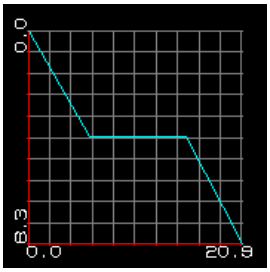
Key Concepts

- You can define pond perimeters from polylines or contours.
- You can import existing pre-defined pond shapes into the drawing.
- You can shape a pond by applying a template to the pond, by defining single or multiple slopes for the pond, or by defining what the final pond volume should be.
- A pond template is a cross-sectional view of the pond perimeter.
- You can use daylighting to match the pond side slopes into the existing ground surface model.
- Refer to the SCS (Soil Conservation Service) TR-55 manual for more information regarding detention pond design.

To design a detention pond

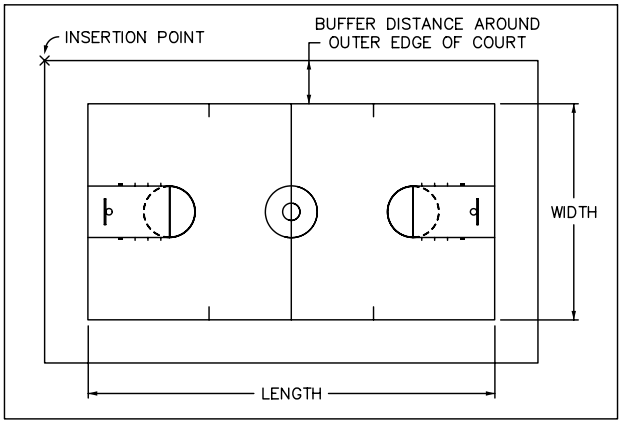
Steps	*Use* Find *to look up*
1 Determine the specific watershed characteristics and design criteria, including the peak flow rate volume to store.	
2 Draw the pond perimeter polyline.	*Draw a Pond Perimeter*
3 From the Grading menu, choose Define Pond ➤ By Polyline to define the pond perimeter polyline.	*Define a Pond Perimeter from a Polyline*

To design a detention pond (*continued*)

Steps	*Use* Find *to look up*
4 From the Grading menu, choose Pond Slopes ➤ Draw Slope Template to draw the pond slope template polyline. There are several ways to shape the pond. One method is to use a pond slope template, as shown below.  The pond slope template is essentially a cross section view of the pond perimeter. You draw the pond slope template at a 1:1 scale, and then you can apply it to the pond perimeter.	*Draw a Pond Slope Template*
5 To define the pond template, from the Grading menu, choose Pond Slopes ➤ Define Template.	*Define a Pond Slope Template*
6 To designate the current template, from the Grading menu, choose Pond Slopes ➤ Set Current.	*Select the Current Pond Slope Template*
7 From the Grading menu, choose Pond Slopes ➤ By Template to apply the current pond slope template to all the vertices of the pond perimeter polyline.	*Apply a Slope Template to a Pond*
8 Type **Yes** when you are prompted to Shape Pond. Shaping the pond brings pond slope data and contours into the drawing.	
9 Verify that the detention pond design meets the design criteria and conditions.	

Adding Landscape Symbols to Drawings

To put the finishing touches on your finished ground site, you can add symbols to depict various sports fields, patios and walks, and parking lots. The following illustration is an example of a basketball court symbol.



Key Concepts

- You can move, scale, and rotate the symbols after you insert them using the grip editing commands.
- You can insert walkways and patios with various paving styles and hatch patterns.
- You can create a parking lot design with a variety of spacing options, such as for handicap access.
- You can insert layout symbols for various sports and activities such as tennis and basketball courts, football and soccer fields, baseball diamonds, and running tracks.

For more information about creating landscape details, use Find to look up "Overview of Creating Track and Field Elements" and "Overview of Creating Walks and Patios" in the online Help.

3

Performing Hydrologic Studies

Autodesk Civil Design provides a variety of methods you can use to calculate runoff from a site, perform routing, and design detention basin inflow and outflow structures.



In this chapter

- Overview of Hydrologic Studies
- Gathering Data for Hydrologic Analysis
- Using the Hydrology Calculators
- Using the Culvert Calculator
- Using the Rational Method to Calculate Runoff
- Using the TR-55 Graphical Peak Discharge Method to Calculate Runoff
- Using the TR-55 Tabular Hydrograph Method to Calculate Runoff
- Estimating TR-55 Detention Basin Storage

Overview of Hydrologic Studies

Early in the process of evaluating a site, you must evaluate how your proposed development will affect watershed runoff. In general, most urban and rural developments alter the hydrological character of a site by reducing the pervious surface area, which ultimately decreases infiltration and travel times.

Since the amount of runoff is directly related to the infiltration characteristics of the site, any development which decreases the pervious surface area adversely changes the watershed's runoff response to precipitation resulting in higher peak discharges. In addition, decreasing travel times causes the peak discharge to occur earlier in the storm water event. To evaluate the impact on the watershed runoff, you can establish pre-development and post-development runoff models, and then compare the results.

For example, it is commonly a requirement of most reviewing agencies that post-development discharges do not exceed pre-development discharges for one or more storm frequencies. To control post-development peak discharges, you can calculate the required storage volume for one or more selected storm frequencies, and then design a detention pond to accommodate increases in storm water runoff for the selected storm events.

You can use the hydrology commands to:

- Calculate runoff from watershed areas using the Rational, the TR-55 Graphical Peak Discharge and Tabular Hydrograph Methods, and the TR-20 method
- Develop pre- and post-development runoff models
- Design various types of water-retention structures to store excess runoff
- Design and analyze hydraulic conveyance structures such as channels, culverts, and weirs

For example, if you are building a shopping center with a large parking lot that covers existing pervious sandy ground, you can use Autodesk Civil Design to ascertain how the impervious surface area of the parking lot will affect the water runoff. You may decide that drainage culverts that lead to a detention pond may be the best way to prevent flooding problems. You can calculate the type of culverts needed to convey the excess runoff from the parking lot area to an appropriately sized detention pond, including the necessary outlet structures to control discharge to pre-development levels.

Gathering Data for Hydrologic Analysis

When evaluating a site to determine whether development is feasible, you must consider what effect the development of the site will have on the area's runoff amounts. The first step in this process is to gather hydrological data about the site, primarily for the pre-development model. You must have an existing ground surface, and you must know the soil type and current land use of the site.

You can start the watershed hydrologic analysis by using the Terrain ➤ Terrain Model Explorer, located in the AutoCAD Land Development Desktop, to create an existing ground surface model of the site. Then, you can use the watershed command (also within the Terrain Model Explorer) to create polylines that outline the principal watershed areas on the surface model. Later, you can select these polylines when prompted to choose a watershed area when using the Hydrology commands. Soil type information, including soil boundary information, can also be added to your surface model.

Key Concepts

- Before starting a hydrologic analysis of a site, determine the hydrologic soil groups existent at the site, the cover type, treatment, and hydrologic condition. These features will affect the results of the pre-development runoff calculations.
- A good way to start the hydrologic analysis of a site is to use the Terrain Model Explorer to create a surface model, complete with topographical information, watershed boundaries, subarea flow paths, slope arrows, and relevant hydrologic data.
- Your compiled topographic and hydrologic data should extend sufficiently off-site to provide adequate coverage of the drainage area affected by your proposed development.

To add watershed and drainage data to your drawing

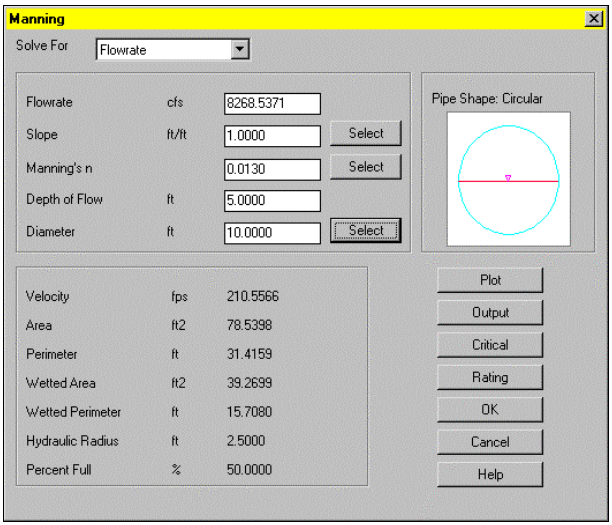
Steps	*Use* Find *to look up*
1 Create an existing ground surface for the proposed site.	*Overview of Creating Surfaces*
2 Generate watershed data for the existing ground surface model.	*Create a Watershed Model After Building the Surface*
3 From the Terrain menu, choose Surface Display ➤ Slope Arrows to draw arrows that follow the slope of the existing surface.	
4 From the Terrain menu, choose Surface Utilities ➤ Water Drop to draw flow paths. The Water Drop command traces the path of a drop of water that lands on the point you pick in the drawing to the point that it will outflow. This can help you determine where the major outflow points are and where you may need to add culverts.	*Draw Water Drop Paths on the Current Surface*

This data can help you visualize the slopes of a surface, where the water will flow, and where the water will accumulate during a storm. You can use this data to decide the best way of controlling the flow. Now that you've visualized the runoff paths on your surface, you can calculate the peak runoff flow for different storm events.

Using the Hydrology Calculators

Many of the features in the Hydrology menu use calculator-type dialog boxes to solve for an unknown value. For each calculator, you must enter the known values in the appropriate edit field for the particular value, or use the corresponding Select button to pick the value from the drawing or from another dialog box. You can select the unknown value that you want to solve for from a popup list at the top of the calculator. If you do not enter all values, then the calculation will not be completed. An error message is displayed at the bottom of the dialog box whenever you make an error entering data.

The following illustration shows a Manning's n gravity pipe calculator. To solve for the flowrate, you enter values in the Slope, Manning's n, Depth of Flow, and Diameter boxes.



You can enter values as mathematical equations. For example, if the required diameter is 36 inches and the required flow percentage in a particular channel is 75%, then enter **36*0.75**, and the value 27.0 is displayed. You can also specify the value in any units and the value will be converted automatically to units that are specified in the settings. For example, if the units are in inches, enter **2"** and the value 24 will be displayed.

Hydraulic structure calculators in Autodesk Civil Design include:

- Darcy-Weisbach pressure pipe
- Hazen-Williams pressure pipe
- Manning's n gravity pipe
- Channel
- Orifice
- Weir
- Riser
- Culvert

Hydrology calculators in Autodesk Civil Design include:

- Time of Travel
- Time of Concentration
- Runoff (Rational, TR-55 graphical and tabular, and TR-20)

Using the Culvert Calculator

Autodesk Civil Design has several features that you can use to design storm water conveyance facilities for controlling the runoff on a site. For example, you can design outlet and inlet structures including channels, culverts, weirs, risers, gravity pipes, orifices, and so on. This section describes how to use the Culvert Calculator to design a culvert. A culvert can be used to channel peak flow amounts under roadways and other structures. You can use slope arrows and water drop trails to determine where the runoff is most likely to cross an alignment. Then you can place culverts at these critical locations.

Key Concepts

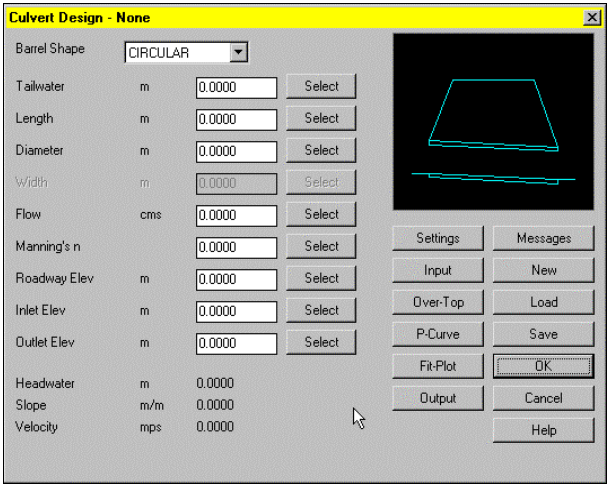
- Determine the peak discharge inflow amount that the culvert has to channel using the Rational Method, the Graphical Peak Discharge Method, the Tabular Hydrograph Method, or an inflow hydrograph
- Consider outlet and tailwater control conditions
- Consider entrance and exit loss conditions
- Consider over-topping conditions
- Consider minimum and maximum design flow velocities to prevent the effects of scouring or related erosion problems

To design a culvert

Steps	*Use* Find *to look up*
1 Determine the specific watershed characteristics and design criteria, including the peak flow rate amounts at the discharge point.	
2 From the Hydrology menu, choose Settings to display the Hydrology Tools Settings dialog box.	*Change the Hydrology Unit Settings*
3 Click Units to specify the culvert measurement units or click Precision to specify the required precision settings for your units.	

To design a culvert (*continued*)

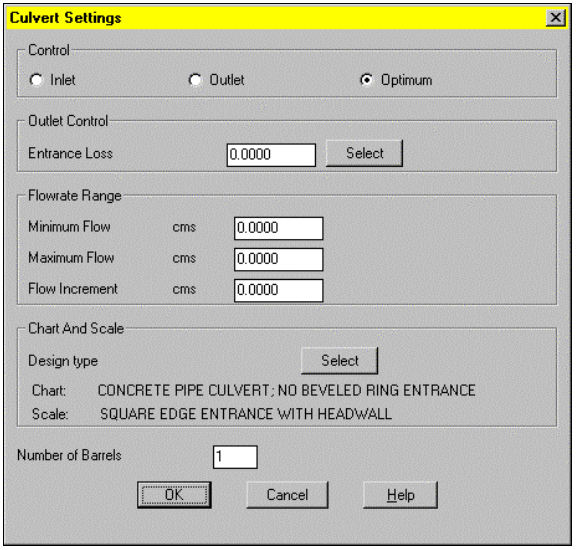
Steps	*Use* Find *to look up*
4 From the Hydrology menu, choose Culvert Calculator to display the Culvert Design dialog box.	*Calculate Culvert Size and Shape*



5 Select the applicable barrel shape from the list. You can select circular or box for the shape of the barrel.	
6 Specify the tailwater length. You can type a value for the tailwater, or you can click Select to display the Tailwater Editor dialog box.	*Specify the Tailwater Length*
7 Specify the culvert length and diameter for a circular barrel, or the width and height for a box barrel. You can type values for these parameters, or you can choose the Select buttons and pick points in your drawing.	*Specify the Culvert Length* *Specify the Culvert Diameter*

To design a culvert (*continued*)

Steps	*Use* Find *to look up*
8 Specify the flow rate for the culvert. You can type a value directly, or you can calculate a flow rate value by clicking Select to display the Runoff Editor dialog box. From here, you can display the Runoff Method Selection dialog box to select an appropriate runoff method where you can then either import or calculate the flow.	*Specify the Flowrate for a Culvert*
9 Specify the Manning's n roughness coefficient value for the culvert. You can type a value for Manning's n, or you can click Select and pick a Manning's n value from a list of standard values based on different types of culvert materials.	*Specify a Manning's n Roughness Coefficient*
10 Specify the roadway elevation, the culvert inlet elevation, and the culvert outlet elevation.	
11 Click the Settings button to display the Culvert Settings dialog box to specify inlet, outlet, or optimum control conditions, entrance losses, flow rate ranges, and number of culvert barrels.	*Change the Culvert Settings*

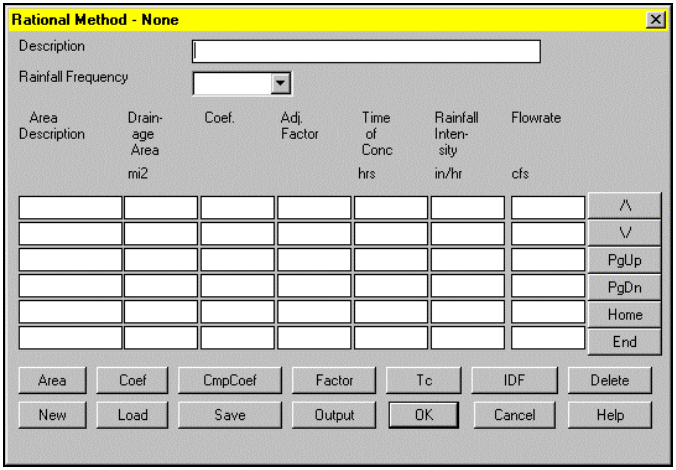


To design a culvert (*continued*)

Steps	*Use* Find *to look up*
12 Click OK to close the Culvert Settings dialog box and return to the Culvert Calculator.	
13 Click Over-Top to access the Culvert Weir Editor dialog box to check the overtop conditions of the culvert.	*Change the Overtop Flow Values to Use in the Culvert Calculations*
14 Verify that the culvert design meets all of the relevant design criteria and conditions.	
15 Create a Performance Curve graph for the designed culvert by clicking P-Curve.	*Display a Performance Curve for a Culvert*
16 Create a Fit Curve graph for the designed culvert by clicking Fit-Plot.	*Display a Headwater Versus Flow Curve for a Culvert*
17 Click the Save button to save your culvert design data to a file.	
18 Click OK to exit the Culvert Design calculator.	

Using the Rational Method to Calculate Runoff

Autodesk Civil Design provides several different methods for calculating peak runoff from a watershed area. One of these methods is the Rational Method. Despite its many governing limitations, the Rational Method still remains the most widely used method for calculating storm water runoff in small urban areas or for highway drainage. The method is based entirely upon a rational analysis of the rainfall-runoff process in which a simple formula, Q = CIA, is used to estimate the peak runoff occurring in the defined watershed area for the selected storm event. This estimate of peak runoff can then be used as a design flow for sizing proposed inlets, pipes, culverts and other hydraulic structures.



Key Concepts

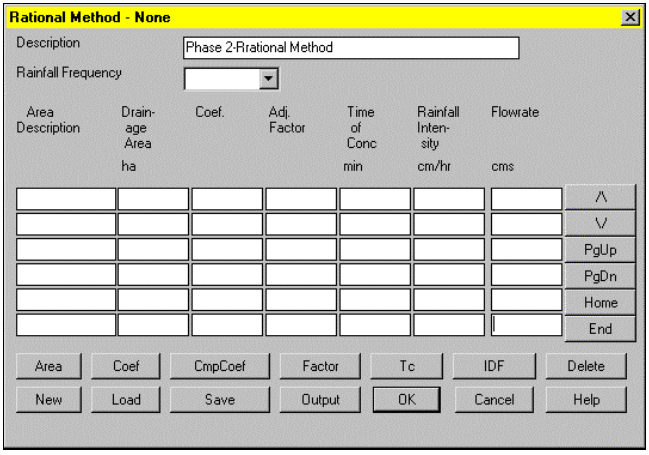
- Establish an intensity duration frequency (IDF) curve file (.idf extension) for your project location.
- Determine the size of the drainage area (A), the runoff coefficient (C), the adjustment factor, the time of concentration (Tc), the rainfall frequency, and the rainfall intensity. This can all be calculated or selected using commands from the Hydrology menu.
- Slopes and elevations across a site can be extracted from a surface model. You can also build a surface and model the watershed before calculating runoff by using the AutoCAD Land Development Desktop Terrain Model Explorer.
- Refer to the AASHTO (American Association of State Highway and Transportation Officials) Model Drainage manual for more information regarding the Rational Method.

To calculate the peak discharge using the Rational Method

Steps	*Use* Find *to look up*
1 Determine the specific watershed characteristics and design criteria, including watershed location/area, soil type, land use, and sheet, shallow, and channel flow parameters. Create your intensity duration frequency (IDF) curve file (.idf extension) from applicable rainfall data for your project location.	

To calculate the peak discharge using the Rational Method (*continued*)

Steps	*Use* Find *to look up*
2 From the Hydrology menu, choose Settings to display the Hydrology Tools Settings dialog box.	*Change the Hydrology Unit Settings*
3 Click the Units button to specify the measurement units. Click Precision to specify the required precision settings for your units.	
4 From the Hydrology menu, choose Runoff ➤ Rational to display the Rational Method dialog box.	*Calculate the Peak Runoff Flow for an Area by Using the Rational Method*



5 Click the IDF button to display the Intensity-Frequency Factor Editor. Select your IDF curve file. From the editor, click the Load button to load your IDF curve file for the project area, and then click OK to return to the Rational Method dialog box.	*Specify the Rainfall Intensity*
6 Select the applicable rainfall frequency from the popup list.	
7 Specify the watershed area. You can type a value for the area in the edit box, or, if you created a watershed with the Terrain Model Explorer, you can select the polyline from your drawing by clicking Area and selecting the polyline. You can also draw a new polyline for selection.	*Calculate the Peak Runoff Flow for an Area by Using the Rational Method*

To calculate the peak discharge using the Rational Method (*continued*)

Steps	*Use* Find *to look up*
8 Specify the runoff coefficient. You can type in a value for the runoff coefficient that represents the ratio of runoff to rainfall, or click Coef to select a single value from a list of standard runoff coefficients. You can also click CmpCoef to calculate a composite runoff coefficient value, if applicable, for your site.	*Specify the Rational Runoff Coefficient*
9 Select an adjustment factor. The adjustment factor edit field is not directly editable, but you can click Factor from the Rational Method dialog box to display the Frequency Factor Editor dialog box. Select the Use Frequency Factor check box, and then select the appropriate storm event from the list of events. Click OK to return to the Rational Method dialog box, and add the appropriate adjustment factor for the specified storm event to the adjustment edit field.	*Specify a Frequency Adjustment Factor*
10 Specify the time of concentration value. You can type a value for the time of concentration, or click Tc to display the Time of Concentration Calculator. You can use this calculator to specify the sheet flow, shallow flow, and channel flow parameters and compile the time of concentration data.	*Calculate the Watershed Time of Concentration*
11 Click Save to display the Save Rational Method Data dialog box. Enter the file name and click Save to return to the Rational Method dialog box.	
12 Click OK when you are finished to close the Rational Method dialog box.	

To determine the runoff peak discharge for other storm events, select the new storm frequency from the popup list in the Rational Method dialog box. The software automatically re-calculates the appropriate rainfall intensity and the runoff peak discharge.

For example, if you select 100 from the Rainfall Frequency popup list, the runoff peak discharge for the 100-year storm event is calculated.

Using the TR-55 Graphical Peak Discharge Method to Calculate Runoff

Technical Release 55 (TR-55), prepared by the Soil Conservation Service (SCS), presents two simplified methods for estimating storm water runoff from urbanizing watersheds. Although the procedures found in TR-55 are particularly well suited to urban and urbanizing watersheds, the methods can be applied, in general, to any small watershed when the governing limitations of either method have been adequately addressed.

The simpler of the two methods is the Graphical Peak Discharge Method (GPDM). The Graphical Peak Discharge Method is intended for use on hydrologically homogeneous watersheds for which land use, soils, and cover type are uniformly distributed throughout the watershed. The TR-55 Graphical Peak Discharge Method, as the name of the method implies, determines the peak discharge only. If the watershed in question is heterogeneous, or if hydrographs are required, the TR-55 Tabular Hydrograph Method should be used.

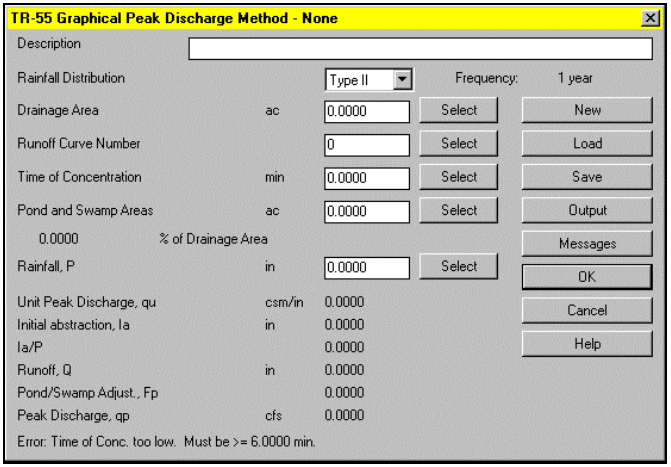
NOTE The Soil Conservation Service is now called Natural Resources Conservation Service.

Key Concepts

- Determine the applicable rainfall distribution type, the size of the drainage area, the runoff curve number (RCN), the time of concentration (Tc), the size of the pond and swamp area, and the amount of rainfall. This can all be calculated or selected using commands from the Hydrology menu.
- Slopes and elevations across a site can be extracted from a surface model. You can also build a surface and model the watershed before calculating runoff by using the AutoCAD Land Development Desktop Terrain Model Explorer.
- Refer to the SCS (Soil Conservation Service) TR-55 manual for more information regarding the Graphical Peak Discharge Method, particularly the implied limitations of the method.

To calculate the peak discharge using the TR- 55 Graphical Peak Discharge Method

Steps	*Use* Find *to look up*
1 Determine the specific watershed characteristics and design criteria, including soil and vegetation types, rainfall frequency and distribution, and sheet, shallow, and channel flow parameters. To do this, use hydrologic data from soil maps, rainfall frequency distribution charts and other relevant publications acquired for your county or region from your local SCS office or county Soil & Water Conservation District office.	
2 From the Hydrology menu, choose Settings to access the Hydrology Tools Settings dialog box. Click Units to specify the Graphical Peak Discharge Method measurement units. Click Precision to specify the required precision settings for your units.	*Change the Hydrology Unit Settings*
3 From the Hydrology menu, choose Runoff ➤ TR-55 Graphical Method to display the TR-55 Graphical Peak Discharge Method dialog box.	*Calculate the Peak Runoff Flow by Using the TR-55 Graphical Method*



4 Select the applicable rainfall distribution type from the Rainfall Distribution list.	

To calculate the peak discharge using the TR- 55 Graphical Peak Discharge Method (*continued*)

Steps	*Use* Find *to look up*
5 Specify the watershed area. You can type a value for the area in the edit box, or, if you created a watershed with the Terrain Explorer, you can select the polyline from your drawing by clicking Select and selecting the polyline. You can also draw a new polyline for selection.	
6 Specify the runoff curve number. You can type in a value for the runoff curve number that represents the hydrological character of your site. You can also click Select to display the Runoff Curve Number Editor dialog box which lists runoff curve numbers based on soil type and surface cover. These values are from Table 2-2 of the TR-55 publication.	*Specify a Runoff Curve Number*
7 Specify the time of concentration value. You can type in a value for the time of concentration, or click Select to display the Time of Concentration Calculator. Use this calculator to specify the sheet flow, shallow flow, and channel flow parameters, and compile the time of concentration data.	*Calculate the Watershed Time of Concentration*
8 Specify the pond and swamp areas adjustment factor. You can type a value for the pond and swamp areas adjustment factor, or click Select, and select one or more closed polyline(s) from your drawing that represent the ponds and swamps in the watershed area.	
9 Specify the 24-hour rainfall amount. You can type a value for the 24-hour rainfall amount, or click Select to display the Define Rainfall Frequency dialog box. Use this dialog box to select the 24-hour rainfall amount for a specified county and storm frequency (1, 2, 5, 10, 25, 50, or 100 years).	*Use the Rainfall Frequency Editor to View, Edit, and Define Rainfall Frequency*

After you enter all the information, the peak discharge is calculated automatically and is displayed in the Peak Discharge line.

To determine the runoff peak discharge for other storm events, select the new storm frequency using the Select button next to the rainfall edit field in the Graphical Peak Discharge Method dialog box,. The software will automatically re-calculate the appropriate rainfall intensity and the runoff peak discharge.

For example, if you select 100 from the Define Rainfall Frequency dialog box, the runoff peak discharge for the 100-year storm event will be calculated.

Using the TR-55 Tabular Hydrograph Method to Calculate Runoff

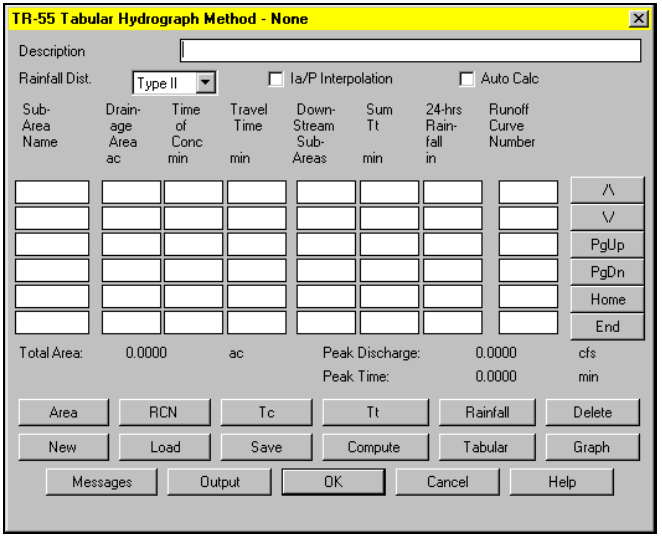
The second runoff procedure outlined in Technical Release 55 (TR-55) is the Tabular Hydrograph Method. The Tabular Hydrograph Method can be used on heterogeneous watersheds that can be subdivided into homogeneous subareas. By dividing the heterogeneous watershed into homogeneous subareas, estimated peak discharges and hydrographs for the heterogeneous watershed can be obtained.

Key Concepts

- Determine the applicable rainfall distribution type for the entire watershed. Additionally, you must know the hydrologic parameters for each subarea, including the size, the time of concentration (Tc), the time of travel (Tt) and, if applicable, the amount of rainfall, and the runoff curve number (RCN). Note that these values can all be calculated or selected from within the Hydrology menu.
- Slopes and elevations across a site can be extracted from a surface model. You can also build a surface and model the watershed before calculating runoff by using the AutoCAD Land Development Desktop Terrain Model Explorer.
- Refer to the SCS (Soil Conservation Service) TR-55 manual for more information regarding the Tabular Hydrograph Method, particularly the implied limitations of the method.

To calculate the peak discharge using the TR-55 Tabular Hydrograph Method

Steps	*Use* Find *to look up*
1 Determine the rainfall distribution type for the specific watershed.	
2 For each subarea, determine the subarea's hydrologic parameters, including the area, time of concentration, travel time, 24-hour rainfall, and runoff curve number. To do this, use hydrologic data from soil maps, rainfall frequency distribution charts and other relevant publications acquired for your county or region from your local SCS office or county Soil & Water Conservation District office.	*Overview of Calculating the Runoff from the Watershed Areas*
3 From the Hydrology menu, choose Settings to access the Hydrology Tools Settings dialog box. Click Units to specify the TR-55 Tabular Hydrograph Method measurement units. Click Precision to specify the required precision settings for your units.	*Change the Hydrology Unit Settings*
4 From the Hydrology menu, choose Runoff ➤ Tabular to display the TR-55 Tabular Hydrograph Method dialog box.	*Calculate the Peak Runoff by Using the TR-55 Tabular Method*



To calculate the peak discharge using the TR-55 Tabular Hydrograph Method (*continued*)

Steps	*Use* Find *to look up*
5 Select the rainfall distribution type from the Rainfall Distribution list.	
6 Specify the subarea name.	
7 Specify the area of subarea #1. Type a value for the subarea's area in the edit box, or, if you created a watershed with the Terrain Model Explorer, select the polyline representing the subarea from your drawing by clicking Area and selecting the polyline. You can also draw a new polyline representing the limits of the subarea for selection.	*Calculate the Peak Runoff by Using the TR-55 Tabular Method*
8 Specify the time of concentration value for subarea #1. Type in a value for the time of concentration, or click Tc to display the Time of Concentration Calculator. Use this calculator to specify the sheet flow, shallow flow, and channel flow parameters, and compile the time of concentration data for subarea #1.	*Calculate the Watershed Time of Concentration*
9 Specify the time of travel value for subarea #1. Type in a value for the time of travel, or click Tt to display the Time of Travel Calculator. Calculate the time it takes for runoff from one subarea to travel through another subarea to the composite watershed outflow point.	*Calculate the Watershed Time of Travel*
10 Specify the downstream subareas for subarea #1.	

To calculate the peak discharge using the TR-55 Tabular Hydrograph Method (*continued*)

Steps	*Use* Find *to look up*
11 Specify the 24-hour rainfall amount for subarea #1. Type a value for the 24-hour rainfall, or click Rainfall to display the Define Rainfall Frequency dialog box. You can use this dialog box to select the 24-hour rainfall amount for a specified county and storm frequency (1, 2, 5, 10, 25, 50, 100).	*Select the Rainfall Frequency for a County*
12 Specify the runoff curve number for subarea #1. Type in a value for the runoff curve number that represents the hydrological character of your subarea, or you can click RCN to display the Runoff Curve Number editor. The Runoff Curve Number editor lists runoff curve numbers based on soil type and surface cover. These values are from Table 2-2 in the TR-55 manual.	*Select and Edit the Runoff Curve Numbers for Different Soil Groups and Cover Types*
13 Specify the hydrologic data for the remaining subareas.	
14 Click Compute to calculate the Peak Discharge and Peak Time values. You can click Graph to create a hydrograph from the current data. Normally, you would compile the pre-development TR-55 Tabular Hydrograph Method data for your project site using the above steps, and then repeat the above steps modifying the hydrological values for the site as required to compile the post-development TR-55 Tabular Hydrograph Method data.	*Overview of Outputting Hydrology Data*
15 Compare the pre- and post-development TR-55 Tabular Hydrograph Method data to assess the impact of site development on the watershed.	

Estimating TR-55 Detention Basin Storage

Typically, most agencies, charged with reviewing storm water management plans for developing sites, require that post-development discharges from the site are equal to or less than pre-development discharges for one or more storm frequencies. To meet this governing requirement, most designers generally employ detention type facilities in strategic locations across the site. The detention basin is generally the least expensive and most reliable measure for controlling post-development peak discharges.

After calculating the peak pre-development outflow and the peak post-development inflow for a site, you can use the TR-55 Detention Basin Storage feature to estimate the storage volume required by your detention pond to control post-development generated runoff.

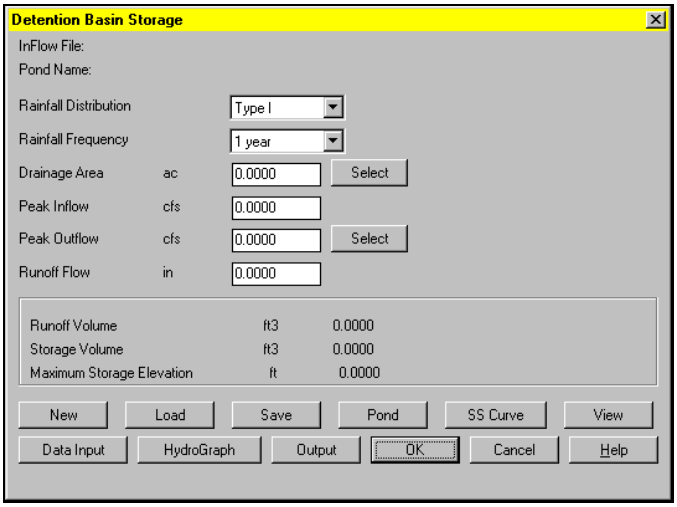
The TR-55 Detention Basin Storage procedure is based on the average storage and routing results obtained from analyzing many detention structures and is biased in favor of oversizing the designed detention facility. The procedure should not be used for final pond sizing design if an error of 25% in calculated storage volume is not acceptable.

Key Concepts

- You can use the runoff methods outlined earlier to determine the peak inflow discharge into the detention pond and the peak outflow discharge from the detention pond.
- You can build a surface and model the watershed before calculating storage requirements by using the AutoCAD Land Development Desktop Terrain Model Explorer.
- You can use existing data when you calculate the required storage volume. Some of the different files you can use are *.tab files generated by the TR-55 Tabular Hydrograph Method, *.ssc stage-storage curve files, *.hdc hydrograph files, and *.bsn files.

To calculate the required storage volume for ponds

Steps	*Use* Find *to look up*
1 Determine the pre- and post-development watershed hydrological characteristics of the site.	
2 Use one of the Hydrology runoff methods described in the preceding topics to determine the post-development peak inflow discharge to the detention basin and the pre-development peak outflow discharge from the detention basin.	*Overview of Calculating the Runoff from the Watershed Areas*
3 From the Hydrology menu, choose Routing ➤ Detention Basin Storage to display the Detention Basin Storage dialog box.	*Calculate the Required Storage Volume for a Detention Basin*



4 Click the Data Input button or the Hydrograph button to load an Inflow file. The InFlow File label displays the name of the currently loaded file that defines the Peak Inflow. This file can be a graphical, tabular, or hydrograph file.	
5 The Pond Name label displays the name of the currently selected pond. Click the Pond button to select an existing pond from the drawing.	

To calculate the required storage volume for ponds (*continued*)

Steps	*Use* Find *to look up*
6 Select the applicable rainfall distribution from the Rainfall Distribution list.	
7 Specify the drainage area. Type a value for the area in the edit box, or, if you created a watershed with the Terrain Model Explorer, select the polyline from your drawing by clicking Select and selecting the polyline. You can also draw a new polyline for selection.	*Specify the Drainage Area*
8 Specify the peak inflow value if you did not load an Inflow file in step 4.	*Load an Existing Hydrograph File*
9 Specify the peak outflow value. Type a value for the peak outflow, or click Select and enter values in the Pond Outflow Design dialog box. You can add outflow structures to control the flow, such as weirs, culverts, or gravity pipes. If you have a defined pond, you can click Pond and choose the pond you want to use for these calculations.	*Specify the Peak Outflow*
10 Specify the runoff flow value.	
11 Click the Save button to display the Save Basin Data dialog box. Enter the file name, and click Save to return to the Detention Basin Storage dialog box.	

When you have entered all the values, the runoff volume and the computed storage volume for the detention basin is displayed at the bottom of the dialog box. If you have a currently defined pond, then the maximum storage elevation for the currently defined ponds is listed as well.

4

Creating Plan Details



In this chapter

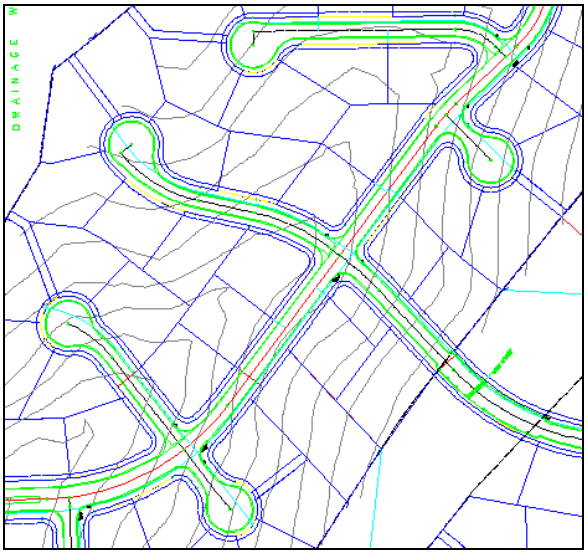
- Overview of Creating Plan Details
- Creating Intersections
- Creating Cul-de-Sacs

You can use the Layout commands to design cul-de-sacs and intersections for plan alignments.

Overview of Creating Plan Details

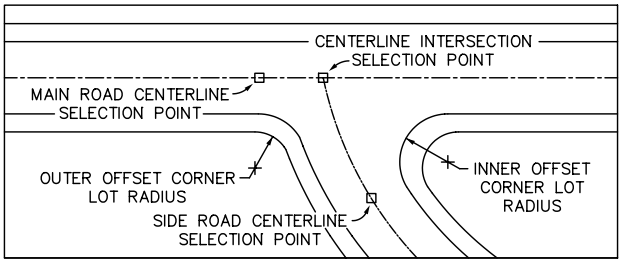
The AutoCAD Land Development Desktop contains a full set of commands that you can use to draw and define road alignments. After you have created and defined an alignment, you can use the Autodesk Civil Design commands to add the finishing touches to the alignment, such as cul-de-sacs and intersections.

For example, to create the subdivision design as shown below, you could offset the centerline alignments, and then create cul-de-sacs at the alignment ends. You can also clean up the intersections where the multiple alignments and their alignment offsets meet.



Creating Intersections

You can use the Autodesk Civil Design intersection commands to easily clean up lines where road alignments cross. The intersection commands automate the process of intersection creation, breaking lines where necessary, and filleting curves. The following illustration shows the intersection of two tangents.



Key Concepts

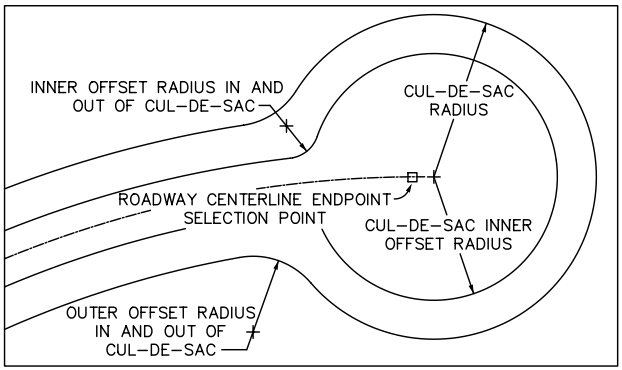
- Use continuous line types when you are designing alignments that will meet in intersections.
- You can use AutoCAD commands like BREAK, TRIM, and FILLET to create intersections if you do not want to use the automated Intersection commands.
- You can manually place points along the intersection geometry using commands in the Points menu in order to create stakeout reports.

To design intersections

Steps	*Use* Find *to look up*
1 Draw the roadway centerline alignments for the intersection by selecting commands from the AutoCAD Land Development Desktop Lines/Curves menu. Or, draw the roadway centerlines using polylines.	*Overview of Lines and Curves*
2 If you drew the alignments with lines and curves, from the Alignments menu, choose Define From Objects to define the roadway alignments. If you drew the alignments using polylines, from the Alignments menu, choose Define From Polyline to define the alignments.	*Define an Alignment from Objects* *Define a an Alignment from a Polyline*
3 From the Alignments menu, choose Create Offsets to create offsets for the alignments.	*Create Offsets for an Alignment*
4 From the Layout menu, choose Intersection Settings to set the intersection settings.	*Change the Intersection Settings*
5 Select one of the intersection commands from the Layout menu to create the intersection. You can select different intersection commands depending on whether the intersection is made up of curves or tangents, and whether the alignments cross or not. For example, if you are designing an intersection where two tangents cross, from the Layout menu, choose 4 Way Intersection ➤ Tangent-Tangent.	*Overview of Cleaning Up Roadway Intersections*
6 Use the commands from the Points menu to set critical points along the intersection. You can place points along the intersection geometry, such as at the point of curvature.	
7 Generate a stakeout report of the alignment Centerline for the surveyor. From the Alignments menu, choose Stakeout Alignment ➤ Create File.	*Create an Alignment Stakeout Report*

Creating Cul-de-Sacs

Autodesk Civil Design has a set of commands that you can use to design five different types of cul-de-sacs: tangent, curved, hammerhead, elbow, and teardrop. The following illustration shows a cul-de-sac drawn off a curved roadway.



Key Concepts

- You should use continuous line types when you are adding a cul-de-sac to an alignment.
- All cul-de-sac commands treat a single offset as the outer offset.
- The offset widths that you specify in the Cul-de-sac Settings dialog box must match the widths of the alignment offsets that are drawn in the drawing.

To design cul-de-sacs

Steps	*Use* Find *to look up*
1 Draw the roadway centerline alignments for the cul-de-sacs by selecting commands from the AutoCAD Land Development Desktop Lines/Curves menu. Or, you can draw the roadway centerlines using polylines.	*Overview of Lines and Curves.*
2 If you drew the alignments with lines and curves, then from the Alignments menu, choose Define From Objects to define the roadway alignments. If you drew the alignments using polylines, then from the Alignments menu, choose Define From Polyline to define the alignments.	*Define an Alignment from Objects*
3 From the Alignments menu, choose Create Offsets to create offsets for the alignments.	
4 From the Layout menu, choose Cul-De-Sacs ➤ Settings to set the cul-de-sac settings. These settings control radii, offset widths, and offset layers.	*Change the Cul-de-sac Settings*
5 Create the cul-de-sac by selecting one of the Cul-de-sac commands from the Layout ➤ Cul-De-Sacs menu.	*Overview of Creating Cul-de-sacs*

5

Viewing and Editing Roads in Profile View

If you have a plan alignment and an existing ground surface, you can generate a profile of the roadway that you can use to design the finished ground alignment.



In this chapter

- Overview of Viewing and Editing Roads in Profile View
- Creating Existing Ground Profiles
- Creating Finished Ground Road Profiles
- Editing Vertical Alignments

Overview of Viewing and Editing Roads in Profile View

After you draft and define a horizontal alignment for a road, you can create a road profile (also known as a vertical alignment or long section) that represents the existing and finished grades along the roadway centerline. To work in profile view, start by creating an existing ground profile for a defined alignment by sampling elevation data from a surface. You can then draft the existing ground profile in the drawing, and draw the vertical alignments and vertical curves that represent the finished ground profile design.

The finished ground profile commands are divided into the following two sets of commands:

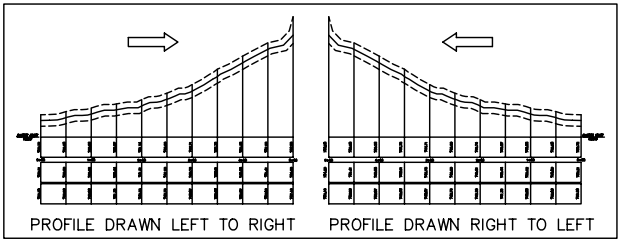
- Finished ground centerline commands: use these commands for drawing and defining the roadway centerline in profile view
- Ditches and transitions commands: use these commands for drawing and defining vertical offsets, such as ditches and transition lanes

When you are drafting and defining vertical alignments, you must select the command from the appropriate menu selection for the type of vertical alignment you are creating.

After you draw a vertical alignment, you must define it as you do with horizontal alignments. The finished ground elevations are used later for calculating the elevations for the roadway cross sections.

Creating Existing Ground Profiles

You can draft an existing ground profile in your drawing and then add vertical alignment geometry to represent what the final roadway will look like in profile view. The following illustration shows existing ground profiles drawn in different directions.



To generate the station/elevation information required to plot a profile, you can extract data from a surface or from an ASCII text file, or you can type in station/elevation values using the profile editor.

Key Concepts

- When sampling the profile from a surface model, be certain that the correct surface model is set current.
- Verify that the existing ground surface model is accurate. Create a model that best reflects the conditions on the site.
- You can set independent scales for horizontal and vertical features. Make sure that the vertical scale is set properly for your drawing.
- A profile has an invisible block attached to it to locate it in the drawing. If you move the profile, first undefine the profile to remove the old profile definition block, then redefine the profile to create a new profile definition block. These commands are in the Profiles ➤ Create Profile menu.
- If you have more than one profile in a drawing, then use the Set Current Profile command to select the correct profile to use in subsequent profile commands. This is a graphical method of selecting the current profile by picking a location within the profile.

To create an existing ground profile

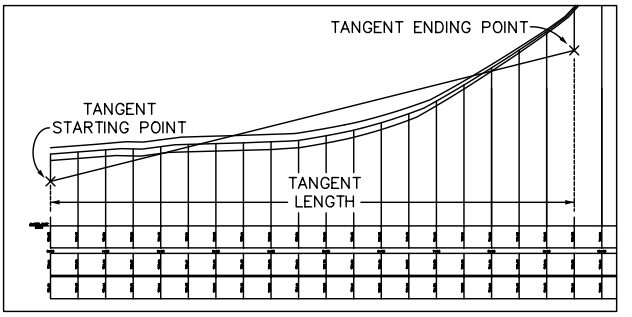
Steps	*Use* Find *to look up*
1 From the Alignments menu, choose Select Current Alignment to make sure that the proper alignment is set as current.	*Make an Alignment Current*
2 Sample the existing ground data (either from a terrain model surface, an ASCII text file, or manual input) by using one of the commands in the Profiles ➤ Existing Ground menu.	*Overview of Sampling the Existing Ground to Create the Profile Data*
3 From the Profiles menu, choose Create Profile ➤ Full Profile to draft the profile. The profile can be drawn from either left to right or right to left. You can also control the profile datum, scale, and use of a grid. You can draw the entire profile at one time or you can import stages of the alignment.	*Create a Complete Profile*
4 You can add a grid to the profile any time after the profile is drawn. To add a grid, from the Profile menu, choose Create Profile ➤ Grid.	*Draw a Grid on a Profile*

Creating Finished Ground Road Profiles

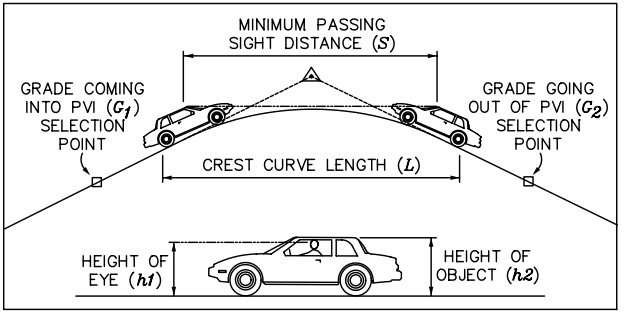
After you create an existing ground profile, you can draw the proposed finished ground profile elements, including the finished ground centerline, offsets, and ditches and transitions.

The profile view of the roadway geometry is referred to as a “vertical alignment.” Vertical alignments are composed of vertical tangents and vertical curves.

The following illustration shows a vertical tangent.



The following illustration shows a vertical curve based on passing sight distance.



Key Concepts

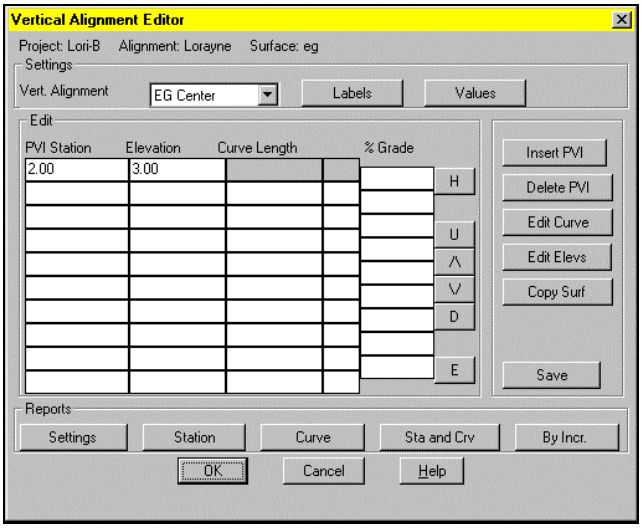
- In addition to the finished ground profile, you can design ditches and transitions in profile view.
- You can use the Create Tangents commands on the Profile menu, or the AutoCAD LINE command to draw vertical tangents, but you must use the Vertical Curves commands to draw vertical curves.
- Other useful tools for drafting vertical tangents are available from the Profiles ➤ FG Centerline Tangents menu.
- In order to properly define the finished ground profiles, you must draw them on the correct layer. Before drawing any entities, set the current layer with the Set Current Layer command.
- After you design finished ground elements in profile view for transition control and ditches, you can "attach" them to the cross sections, automatically updating the templates with the ditch and transition elevations you established in profile view.

To create a finished ground profile centerline

Steps	*Use* Find *to look up*
1 Draft the existing ground profile.	*Overview of Creating Profiles*
2 From the Profiles menu, choose FG Centerline Tangents ➤ Set Current Layer to set the current layer.	*Set the Current Layer for the Finished Ground Profile Centerline*
3 From the Profiles menu, choose FG Centerline Tangents ➤ Create Tangents to draw proposed tangents based on stations, elevations, lengths, and grades. You can adjust the AutoCAD crosshairs to a selected grade if needed. To adjust the crosshairs, from the Profiles menu, choose FG Centerline Tangents ➤ Crosshairs @ Grade. This command affects the AutoCAD snap angle variable and turns ortho mode on. Remember, the vertical scale is exaggerated. Autodesk Civil Design automatically factors in this scale exaggeration.	*Draw the Vertical Alignment Tangents for the Finished Ground Centerline*
4 From the Profiles menu, choose FG Vertical Alignments ➤ Define FG Centerline to define the finished ground centerline. When you select this command, all of the layers other than the FG Centerline layer will be turned off so you can quickly select only the FG Centerline objects.	*Define the Finished Ground Centerline as a Vertical Alignment*

Editing Vertical Alignments

You can edit any existing ground or finished ground vertical alignment using a tabular editor called the Vertical Alignment Editor. If you have sampled the existing ground surface, then you can use this editor to view or edit the information that was generated. You can also use this editor to create existing ground or finished ground information.



Key Concepts

- You can use the Vertical Alignment Editor to create and edit a vertical alignment, to edit vertical curves, to copy vertical alignments, to edit profile elevations, and to generate vertical alignment reports.
- The Vertical Alignment Editor is not dynamically linked to the drawing. After you make edits, you must re-import the vertical alignment into the drawing to update the changes.
- If you edit the existing ground profile by using the Vertical Alignment Editor, then you must recreate the profile with the Create Full Profile command.

To edit a vertical alignment

Steps	*Use* Find *to look up*
1 From the Profiles menu, choose Existing Ground ➤ Edit Vertical Alignment to display the Vertical Alignment Editor.	*Create and Edit the Vertical Alignment Data with the Vertical Alignment Editor*
2 Select the vertical alignment that you want to edit from the Vert. Alignment list.	
3 You can edit elevations, points of intersection, and vertical curves.	*Edit a Vertical Curve with the Vertical Alignment Editor*
4 You can generate vertical alignment reports by station, vertical curve, and increments.	
5 When you have finished editing the alignment, close the Vertical Alignment Editor by clicking OK.	
6 From the Profiles menu, choose Vertical Alignments ➤ Import to import the edited proposed profile alignment back into the drawing.	*Import the Ditch or Transition Vertical Alignments*

6

Viewing and Editing Roads in Section View

To design a roadway in cross-sectional view, create a roadway template and apply it to the plan alignment and profiles. When working in section view, you can superelevate and transition the road to meet design requirements.



In this chapter

- Overview of Viewing and Editing Roads in Section View
- Creating Existing Ground Sections Along a Road
- Working with Templates
- Creating Finished Ground Cross Sections
- Editing Cross Sections
- Transitioning a Roadway
- Modifying a Roadway Slope
- Superelevating a Roadway
- Using Roadway Data for Finished Ground Surfaces

Overview of Viewing and Editing Roads in Section View

After you have created an alignment and profile for a roadway, you can generate cross sections. Cross sections are cut at stations along an alignment.

Using the Cross Sections commands, you can:

- Create existing ground cross sections for the alignment
- Create finished ground roadway surface templates
- Establish design parameters for ditches, superelevation, and transitions
- Extract, view, edit, and plot cross sections
- Insert cross sections in a drawing for plotting
- Output volumes using Average End Area or Prismoidal methods
- Place design roadway points in a drawing or external file for field staking
- Create a 3D road grid of the alignment

Creating Existing Ground Sections Along a Road

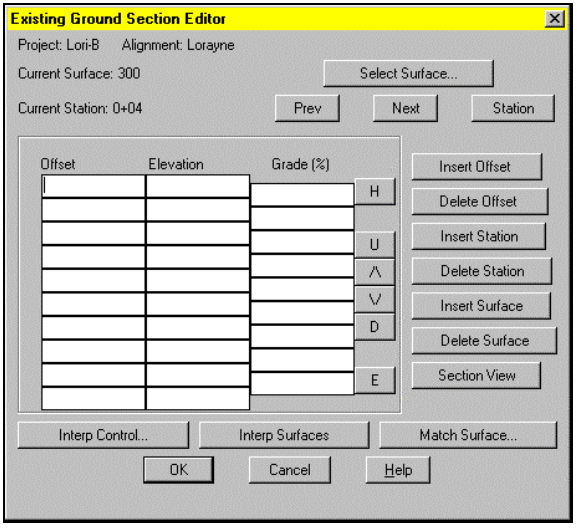
After you have defined the horizontal alignment, you can extract and plot cross sections of the existing ground data.

Key Concepts

- You can extract cross section data from a terrain model or from a station/offset/elevation text file.
- You can plot sections that show existing ground conditions along the roadway.
- To create existing ground cross sections, you must define a road alignment, but a profile is not required. The design profile is required to apply a template to the sections.

To generate existing ground cross sections

Steps	*Use* Find *to look up*
1 From the Alignments menu, choose Set Current Alignment to make sure that the proper alignment is set as current.	*Make an Alignment Current*
2 Generate existing ground section data using one of the commands in the Cross Sections ➤ Existing Ground menu. The data can be extracted from a terrain model, from a station/offset/elevation ASCII text file, or from manual data entry.	*Sample the Existing Ground Section Data from One DTM Surface* *Create the Existing Ground Cross Section Data From a Text File*
3 You can view the cross sections by selecting Cross Sections ➤ View/Edit Sections. Use the Next option to view the cross sections as they progress along the alignment. You can also edit individual cross sections using this command.	*Choose Which Cross Section Station to Edit*
4 From the Cross Sections menu, choose Existing Ground ➤ Edit Sections to edit the cross section data in a tabular editor as shown in the following illustration.	*Edit the Existing Ground Cross Section Data*



To generate existing ground cross sections (*continued*)

Steps	*Use* Find *to look up*
5 You can plot a single section, a page of sections, or all sections by selecting a command from the Cross Sections ➤ Section Plot menu.	*Plot a Single Cross Section*
6 Sections are plotted into the drawing based on the current horizontal and vertical scales.	*Plot Multiple Cross Sections*

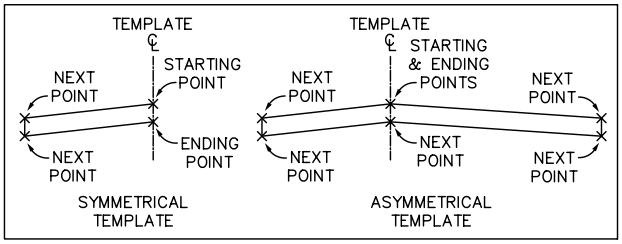
Working with Templates

To create finished ground cross sections, you need to use a design template. A design template represents the road, channel, dam, or railway bed surface and its subsurfaces, such as asphalt, concrete, and granular materials. You can draw the template using an exaggerated scale (based on the drawing's horizontal and vertical scale) so you can better visualize the surfaces. After drawing the template, you define the template and generate the design sections by processing the template. Design sections are generated wherever an existing ground cross section has been sampled.

Key Concepts

- Begin by drawing a template. If the road has the same surface elements on either side, then the template is symmetrical. You only need to draw the left half of a symmetrical template. If the road has one south-bound lane and two north-bound lanes for example, the template is asymmetrical. You must draw both sides of an asymmetrical template.

The following illustration shows the points you need to pick when drawing symmetrical and asymmetrical templates.



- To apply transition and superelevation regions on the template, you must edit the template after you define it.
- Templates can be made up of normal and subgrade surfaces. Normal surfaces are the elements of the template which make up the main part of the template such as pavement surfaces, median islands, shoulders, and curbs. Subgrade surfaces are linked to the normal surfaces, but use separate design parameters to control the grade and depth of the surface. A typical subgrade surface is made up of granular materials, such as gravel, and generally represents materials lying directly over the subgrade (limit of excavation).
- While curbs and shoulders can be defined as part of the template, you can also draw these items separately and define them as subassemblies. Then, when you are defining the template, you can attach the subassembly to the template definition.
- You can use template point codes to insert points into the drawing based on template points, such as the right-of-way and edge-of-pavement.

To work with templates	
Steps	***Use*** Find ***to look up***
1 From the Cross Sections menu, choose Templates ➤ Draw Template to draw the finished ground template.	*Draw a Template Surface - General Procedure*
2 If you want to use a subassembly for a curb or shoulder, then use the Draw Template command to draw the subassembly. From the Cross Sections menu, choose Templates ➤ Define Subassembly to define the subassembly.	*Define a Subassembly*
3 From the Cross Sections menu, choose Templates ➤ Edit Material Table to set up the Material Table. A material table is a collection of surface material names that you can select when you are defining template surfaces.	*Define and Edit a Material Table*
4 From the Cross Sections menu, choose Templates ➤ Define Template to define the template. In this step you define the finished ground reference point, the template geometry, the surface materials, and the depths of subgrade surfaces. You also attach subassemblies (optional) to the template at this point.	*Overview of Defining Templates*
5 From the Cross Sections menu, choose Templates ➤ Edit Template to add transition points and superelevation points to the template if necessary. You can also add top surface points to the template which you can later import into the drawing to use as finished ground data.	*Overview of Editing Templates*

Creating Finished Ground Cross Sections

Before creating finished ground sections, you must:

- Define a road alignment.
- Draw and define a finished ground centerline vertical alignment.
- Have an available road template that you can apply. If no template is available, then you need to draw and define the template.
- Create existing ground cross sections.

The finished ground sections include elevational information, a surface template, slopes, and optional ditches.

To fine-tune the cross sections, you can use the Design Control commands. These commands include options for you to configure slope settings and superelevation, among other options.

Key Concepts

- Each template has a finished ground reference point which is used by the Edit Design Control command to position the template on the cross section using the horizontal alignment and the finished ground vertical alignment for control. The finished ground reference point is usually the crown of the roadway.
- There are two methods that you can use to edit the cross sections after you process them. You can use the Edit Design Control command to edit a range of cross sections, or you can use the View/Edit Sections command to edit individual sections.
- If you want to apply superelevation or transition control to finished ground cross sections, the template must contain transition and superelevation control locations. If that is the case, you can then apply superelevation factors and specify vertical and horizontal transitions when widening or altering the roadways characteristics.
- There are two methods of processing cross sections. If you change any of the cross section design control when you are using the Edit Design Control command, then the sections will be processed automatically as you exit the command. You can also process cross sections manually, from the Cross Sections menu, choose Design Control ➤ Process Sections command.

To create finished ground cross sections

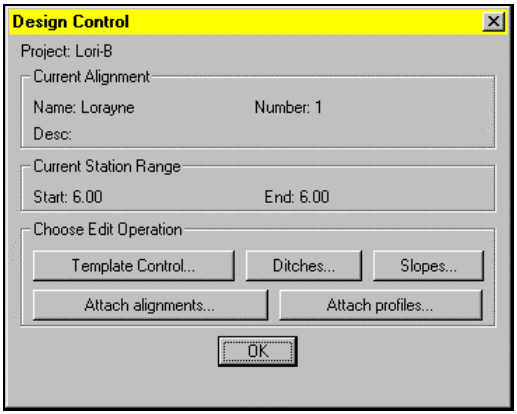
Steps	*Use* Find *to look up*
1 From the Alignments menu, choose Set Current Alignment to make the correct alignment current.	*Make an Alignment Current*
2 From the Profiles menu, choose Set Current Profile to make the correct profile current.	*Make a Profile Current*
3 If you are applying superelevation to the alignment, then set up the superelevation parameters. From the Cross Sections menu, choose Design Control ➤ Superelevation Parameters. NOTE You can set up the superelevation parameters at any time during the design process.	*Change the Superelevation Settings*
4 From the Cross Sections menu, choose Design Control ➤ Edit Design Control to set up the design control parameters and process the sections. These parameters control which template to use when processing cross sections, ditch values, slope control values, transitions, and superelevation. Whenever you modify the design control parameters, the cross sections are processed automatically.	*Use the Edit Design Control Command to Process and Edit the Cross Sections*
5 You can view and edit individual cross sections by selecting Cross Sections ➤ View/Edit Sections.	*Choose which Cross Section Station to Edit*
6 Plot the cross sections using one of the Cross Sections ➤ Section Plot commands.	*Plot a Single Cross Section* *Plot Multiple Cross Sections*

Editing Cross Sections

After you create finished ground cross sections, you can edit the Design Control and re-process a specific range of sections, or all of the sections. You can also edit sections one-by-one if preferred, which is the recommended method for editing the superelevation regions.

Edit Design Control

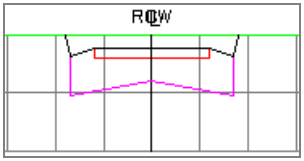
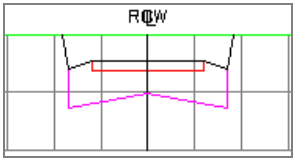
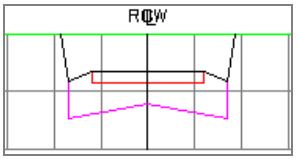
Select Edit Design Control to edit a range of sections. The Design Control dialog box is shown in the following illustration. You can use this command to select which template to use, to define ditches and slopes, and to attach plan and profile alignments to the sections.



View/Edit Sections

Use the View/Edit Sections command to view and edit sections one-by-one. The following illustrations show how sections appear when you use the View/Edit Sections command.

What you see using the View/Edit Sections command

Station 42+00	Station 42+50	Station 43+00
		

The following command prompt is displayed when you use the View/Edit Sections command. You can use the Next, Previous, and Station options to move to a section you want to view or edit.

```
Enter station: 4200
Sta: 42+00   Section View & Display commands
Actual/Design/Edit/Id/Next/Previous/eXit/Sta/View/Zoom <Next>:
```

Edits that you make to individual cross sections with the View/Edit Sections command will not be overridden when you apply different cross section factors to a range of sections with the Edit Design Control command. For example, if you edit the superelevation of three cross sections, and then apply ditch control to the entire range of sections, the superelevation edits you made will not be lost. However, if you edit the superelevation of three cross sections and then apply superelevation parameters to the entire range of cross sections, the edits that you made to the three cross sections will be overridden.



For more information about editing cross sections, use Find to look up “Overview of Using the View/Edit Sections Command to Edit the Cross Sections” in the online Help.

Transitioning a Roadway

To transition a road, you can create plan and profile transition regions on your finished roadway design. For example, if your highway design includes a passing lane on a hill, you can add the additional lane to the plan view of the roadway, define the edge of pavement as a transition alignment, and then update the cross sections using the Edit Design Control command.

You can also design vertical alignments in the profile view that represent vertical transitions, subgrade surfaces, or ditch elevations, and then you can attach these vertical alignments to the cross sections, updating them with the new elevations.

Key Concepts

- In order to create transition regions, you need to define transition control points on the template using the Edit Template command.
- You can create horizontal and vertical transition alignments to attach to the cross sections.
- You can use commands in the Cross Sections ➤ Ditch/Transition menu to define plan and profile transition alignments. However, you can also use commands in the Alignments and Profiles menus to define and edit these transition alignments.
- If you make changes to the transition alignments using the View/Edit Sections command or the Edit Design Control command, then you can use the Cross Sections ➤ Ditch/Transition ➤ Import commands to import these transition alignments back into the plan or profile views.

To transition a roadway

Steps	*Use* Find *to look up*
1 Draw and define the finished ground template. For more information, see "Creating Finished Ground Cross Sections" in this chapter.	*Overview of Defining Templates*
2 From the Cross Sections menu, choose Templates ➤ Edit Template to place transition points on the template. **NOTE** The transition control locations are saved with the template and can be used from one project to the next.	*Define the Transition Regions on a Template*

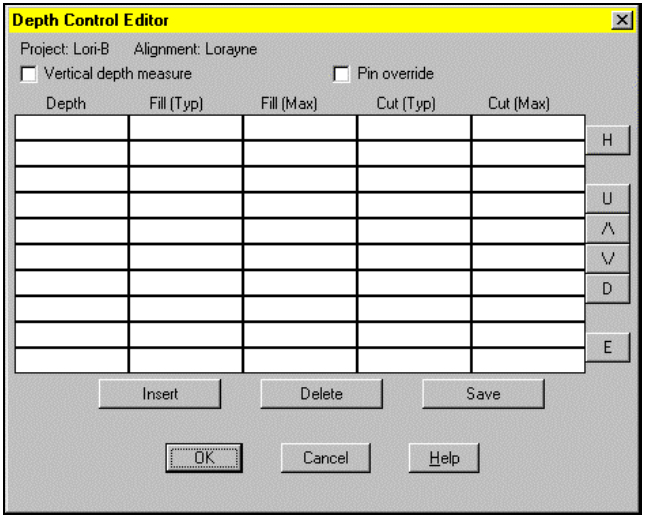
To transition a roadway (*continued*)

Steps	*Use* Find *to look up*
3 Draw and define the horizontal or vertical transition alignments. For example, you can draw a horizontal transition alignment for a passing lane, or a vertical transition alignment for a ditch.	*Define a Ditch or Transition as a Horizontal Alignment* *Define a Ditch or Transition as a Vertical Alignment*
4 From the Cross Sections menu, choose Design Control ➤ Edit Design Control to apply the transition alignments the template. To attach horizontal alignments, click the Attach Alignments button. To attach profiles, click Attach Profiles. When you exit the Edit Design Control dialog box by clicking OK, the cross sections are automatically updated with the transition information.	*Attach Horizontal Transitions to Cross Sections* *Use Ditch or Transition Profiles when Processing the Cross Sections*
5 You can edit individual cross sections if needed using the Cross Sections ➤ View/Edit Sections command.	*Change the Left and Right Transition Regions for One Section*
6 If you want to update the vertical alignment with the edits that you made to the cross sections. From the Cross Sections menu, choose Ditch/Transition ➤ Import Profile to import the transition line into the profile. If you want to update the horizontal alignment with the edits that you made to the cross sections. From the Cross Sections menu, choose Ditch/Transition ➤ Import Plan Lines to import the horizontal transition into the plan view.	*Import a Ditch or Transition from the Sections into a Profile* *Import a Ditch or Transition from the Sections into Plan View*
7 Redefine the imported horizontal and vertical alignments to update the alignment database.	*Overview of Defining Alignments*

Modifying a Roadway Slope

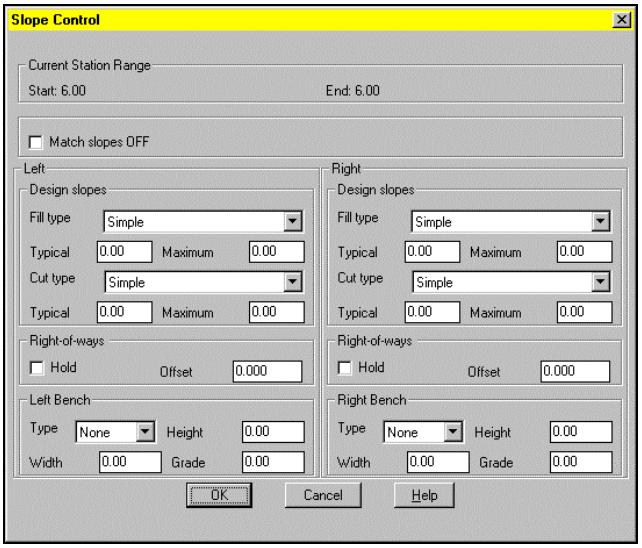
There are several methods that you can use to create match slopes for the cross sections. For each section, you can apply different cut and fill slope conditions to the left and right sides. You can apply simple slopes that follow a linear slope projection (3:1 in cut and 4:1 in fill). You can also specify the use of benching for areas of substantial cut of fill.

There are also more advanced slope calculation methods which vary the design slope based on conditions such as the surface material that you are cutting into and depth of cut/fill. When using these more advanced options, applying slope control to cross sections is a two-step process. First you set up the slope table(s) with the slope values you want to use. The following illustration shows the Depth Control Editor, which you can use to set up depth slope values.



After you set up the slope table(s), you apply these values to the cross sections using the Edit Design Control command.

The following illustration shows the Slope Control dialog box, which you access from the Edit Design Control command.



Key Concepts

- If you just want to use simple slopes, you only need to use the Edit Design Control command. Simple slopes use the typical cut and fill slope values.
- Depth control slopes can use different slopes in cut and fill for various depth ranges based on the depth slope tables that you create from the Cross Sections menu, by choosing Design Control ➤ Depth Slope. With this option, the depth of cut or fill is determined for each section and the appropriate slope is used.
- You can apply benching to simple or depth control slopes based on height criteria. You can define the width and grade of the bench.
- Stepped control slopes are a variation on depth control slopes. Instead of finding the appropriate value for the current depth and applying it as a constant, the slope changes as it passes through each depth range.
- Surface control slopes can be applied in cut situations only and are based on the different existing ground surfaces that they pass through.

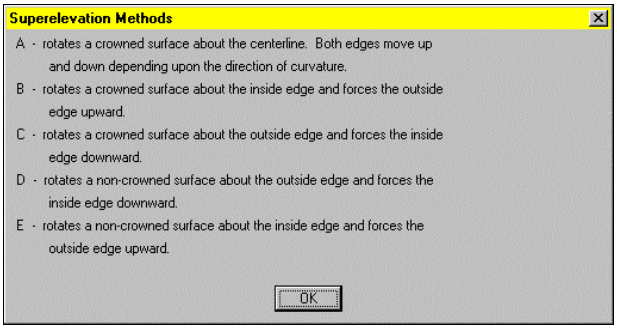
To design slopes for a roadway

Steps	*Use* Find *to look up*
1 Create finished ground cross sections for the roadway. For more information, see "Creating Finished Ground Cross Sections" in this chapter.	
2 If you want to use stepped, surface, or depth control slopes, then you must define the slope tables. Select either Depth Slopes, Stepped Slopes, or Surface Slopes from the Cross Sections ➤ Design Control menu.	*Change the Depth Slope Settings* *Change the Stepped Slope Settings* *Change the Surface Slope Settings*
3 From the Cross Sections menu, choose Design Control ➤ Edit Design Control and then click Slopes to edit the cross section slope control. In this step, you select which type of slope you want to apply in cut and fill situations. When you exit the Slope Control dialog box, the cross sections are processed and updated with the new slope information.	*Specify the Design Control Values for Sideslopes*
4 You can edit the slopes for individual cross sections, if needed, by selecting Cross Sections ➤ View/Edit Sections.	*Change the Slope Control for One Section*

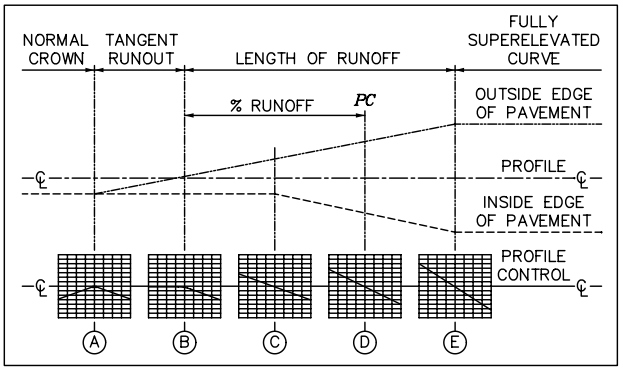
Superelevating a Roadway

You can superelevate a roadway by defining superelevation control points to the roadway template, selecting a superelevation method, and then processing the cross sections.

You can choose one of five superelevation methods for different situations. The following dialog box explains the five methods.



The following illustration shows superelevation method A. The cross sections at the bottom of the illustration show cross sections of the crown at even distances along the profile.



The illustration shows the rate of change in the superelevation is constant between section A and C. The rate of change is constant because the distance between B and C is equal to the distance between A and B (the runout distance). Depending on the design criteria, there may be a change in the rate of change in superelevation at section C.

Key Concepts

- To apply superelevation to cross sections, you need to use the Edit Template command to place superelevation control points on the roadway surface template.

- If you used a speed table to draw the spirals for your alignment, some superelevation information, such as the maximum e value, the runoff lengths, and the percent runoff is already defined for the roadway design.

To superelevate a roadway

Steps	*Use* Find *to look up*
1 From the Cross Sections menu, choose Templates ➤ Edit Template to define the superelevation regions on the finished ground template.	*Define the Template Superelevation Regions on a Template*
2 From the Cross Sections menu, choose Design Control ➤ Edit Design Control and then click Template Control to apply the template to the cross sections.	*Specify the Design Control Values for Templates*
3 From the Cross Sections menu, choose Design Control ➤ Superelevation Parameters to edit the superelevation curve parameters. You can select which method of superelevation to use, edit the subgrade superelevation values, and so on.	*Change the Superelevation Control Values* *Edit, Insert, or Delete a Superelevated Curve*
4 You can generate a report of cross section information by clicking Output in the Superelevation Control dialog box.	*Output the Superelevation Data*
5 From the Cross Sections menu, you can choose View/Edit Sections to view and edit the superelevation at individual cross sections.	*Edit the Superelevation for One Section at a Time*
6 Although profiles don't directly support superelevation, you can convert the superelevation information to a transition so that you can import it into the profile. From the Cross Sections menu, choose Templates ➤ Edit Template to define transition points at the same location as the superelevation points on your template. From the Cross Sections menu, choose Ditch/Transition ➤ Import Profile to import superelevation as a transition line into your profile.	*Import Superelevation into a Profile*

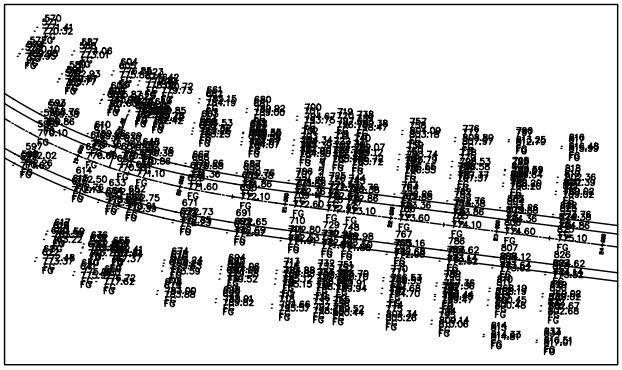
Using Roadway Data for Finished Ground Surfaces

You can place points into a drawing that relate to a finished road design. You can use these points as data for creating a finished ground surface that contains the roadway data.

For example, you can create:

- Existing ground, top surface, and datum template points
- Points based on template point codes
- Catch points and daylight lines

The following illustration shows template points inserted into a drawing.

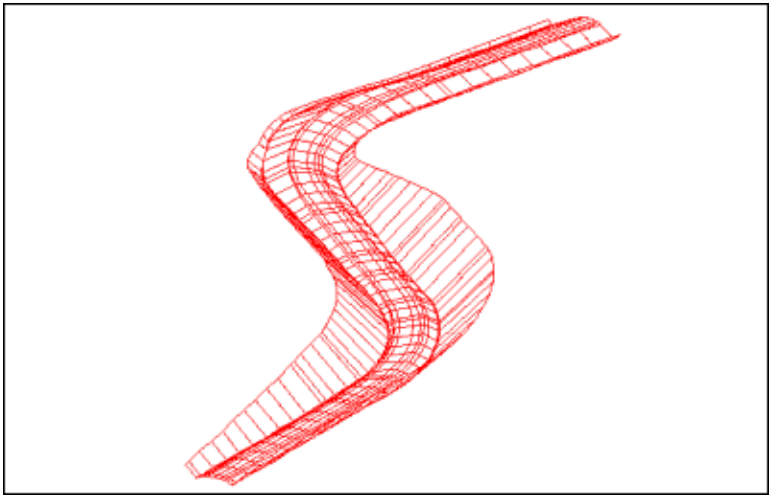


You can process this point data like any other point data and use it to create a finished ground roadway surface. You can then paste this surface into the existing ground surface to create a composite of the two surfaces.

Key Concepts

- If you want to import top surface points, datum points, or custom point codes, then you must first define these points. From the Cross Sections menu, choose Templates ➤ Edit Template, and then reprocess the cross sections.
- When you import top surface data or the datum data into the drawing, both ditch and match slope points will be imported.
- Point codes can include centerline points, ditch points, bench points, catch points, and so on.

You can also create a 3D grid of the roadway by selecting Cross Sections ➤ 3D Grid. Then you can use the point information in that grid for creating the finished ground surface. Process the grid data by using the 3D Faces option. The following illustration shows a 3D grid of a road.



7

Designing Pipe Runs

Begin pipe design by laying out conceptual plan and profile pipe runs. Import finished draft pipe runs to create symbols and labels.



In this chapter

- Overview of Designing Pipe Runs
- Drawing and Defining Pipe Runs
- Importing Plan View Pipe Runs
- Drafting Conceptual Profile Pipe Runs
- Editing Pipes Runs Graphically
- Working with the Pipe Run Editor
- Drafting Finished Plan Pipe Runs
- Drafting Finished Profile Pipe Runs

Overview of Designing Pipe Runs

Autodesk Civil Design has a Pipes menu that you can use to design and draft pipe runs in your drawing that represent either storm water or sanitary sewer collection systems.

You can start by drawing conceptual pipe runs, represented by single lines, or you can import predefined pipe runs into the drawing. You can use terrain models in order to obtain elevational data for the pipe runs and you can associate a pipe run with a roadway alignment for horizontal location data. After you have sized and configured the pipe run, you can draft finished plan and profile pipe runs with a complete feature set of customized labels, node structures, and graphical pipe designations.

You can use the Pipes commands to:

- Design and draft sanitary and storm water sewer systems in both plan and profile views.
- Perform flow, velocity, depth, slope, and other types of analyses to satisfy a variety of design conditions using the Pipes Run Editor.
- Determine hydraulic and energy grade line elevations for your system.
- Size the pipe segments and adjust run variables with the Pipes Run Editor.

Some terms that are referred to in this chapter are described below.

Node: A node is the intersection of individual pipes, or the end of one individual pipe, in a defined pipe run. In a sanitary sewer design, the node is typically represented by a structure such as a manhole.

Pipe: A pipe is the entity that connects two unique nodes.

Run: A pipe run is a collective group of pipes and nodes. A pipe run has a minimum of two nodes connected by a pipe.

Structure: A structure is the physical definition of the node such as a catch basin, manhole, or an item at the end of a pipe.

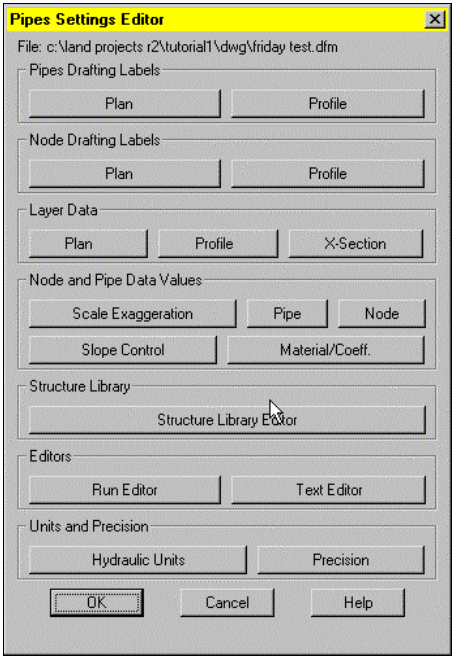
Drawing and Defining Pipe Runs

The first step in designing the pipe run is to lay out the conceptual pipe run in your drawing. Conceptual pipe runs are single line representations of plan and profile view pipe runs. They serve as quick sketches of pipe run configurations, which you can use to check a particular pipe run for proper layout and location.

Key Concepts

- From the Pipes menu, choose Define Pipe Runs ➤ Draw Pipe Run command to draw pipe runs by manually selecting starting and ending points of individual pipe run segments, and specifying their elevations. This command also defines the pipe run to the database.
- From the Pipes menu, choose Define Pipe Runs ➤ Define By Polyline to define the pipe run from an existing polyline in your drawing.
- You can also create a pipe run by importing a file that is saved as an ASCII text file.
- You can draw the pipe run by specifying stations and offsets from an existing alignment.
- You can draw pipe runs with or without referencing a terrain model. A terrain model can provide you with surface elevations for manhole rims, or you can input the manhole elevations manually.
- When you save the pipe run, you can also define the pipe run as an alignment, or you can select an existing alignment to associate the pipe run with. By associating the pipe run with an alignment or by defining it as an alignment, you can draft the pipe run in profile view.
- You can edit various pipe run parameters in the Edit Run Node dialog box, which you can display from the Pipes menu, by choosing Conceptual Plan ➤ Edit Graphical.

To draw and define a pipe run

Steps	*Use* Find *to look up*
1 From the Projects menu, choose Drawing Settings to display the Edit Settings dialog box. Or, from the Pipes menu, choose Settings ➤ Edit to display the Pipes Settings Editor.	*Overview of Changing the Pipe Settings*
2 In the Program list, select Civil Design. In the Settings list, select Pipeworks, then click Edit Settings to display the Pipes Settings Editor dialog box. These settings control the pipe diameter, name, material, coefficient, the formula for calculating pipe flow volume, and the invert depths. 	
3 Click Node to define the node settings. These settings control the node name and structure reference description and node head losses.	*Change the Default Node Data Settings*

To draw and define a pipe run (*continued*)

Steps	*Use* Find *to look up*
4 From the Pipes menu, choose Define Pipe Runs ➤ Draw Pipe Run, and then type a new pipe run name. Select a terrain model (if a surface is defined in your project). You can use this surface to extract rim elevations for the manhole structures located at each pipe run node. You are prompted to turn on or off the current surface. If you want to enter elevations manually, click Off to turn off the surface. If you want to extract elevations from the surface, click On.	*Draw and Define Pipe Runs*
5 If you are basing the run on an existing roadway horizontal alignment, then select an alignment and place the first point of the pipe run by specifying the station and offset from the alignment. If you are drawing the run manually, then specify the first point by picking a point in the drawing or by entering its northing/easting coordinates.	
6 After you specify each point, press ENTER to Add the point to the pipe run.	
7 Type the first point's rim elevation (if it is not being extracted from the current terrain model).	
8 Add the next point by station and offset or by manually picking the point.	
9 Continue adding points in the pipe run.	
10 Type **S** to save your changes to the database. The Run Alignment Association dialog box is displayed.	

To draw and define a pipe run (*continued*)

Steps	*Use* Find *to look up*
11 Select an alignment option. You can create an alignment from the pipe run you just drew, or you can associate the pipe run with an existing alignment or the current alignment. If you select the Create an Alignment from Run option, then you will be prompted to select the alignment start point and the entities that make up the pipe run alignment, just like when you define a roadway alignment. This alignment will be saved to the alignment database. You can use this alignment for drafting the pipe run in profile view.	

Importing Plan View Pipe Runs

You can import existing pipe runs from the pipe database into the drawing. For example, if you delete the entities in your drawing that make up the pipe run, you can import the pipe run back into your drawing. Or, you can import the pipe run into another drawing that is associated with the same project.

Key Concept

- A defined run must exist in the database prior to importing.

To import a plan view pipe run

Steps	*Use* Find *to look up*
1 From the Pipes menu, choose Conceptual Plan ➤ Import Run to display the Defined Runs dialog box.	*Import Conceptual Pipe Runs into Plan View*
2 Select the pipe run that you want to import.	
3 Click OK to import the selected pipe run into the drawing.	

Drafting Conceptual Profile Pipe Runs

You can draft a conceptual pipe run in profile view if you associated the plan pipe run with an alignment or defined an alignment from the pipe run. You can use the conceptual profile view of the pipe run to check for problems with inverts and to make graphical edits to the run in profile view.

Key Concepts

- Draft a profile in your drawing for the alignment that you are associating with the pipe run.
- In order to view and edit the pipe run in your profile, you can import it from the Pipes menu, by choosing Conceptual Profile ➤ Import Run.

To draft a conceptual profile pipe run

Steps	*Use* Find *to look up*
1 Define a conceptual plan pipe run.	*Define Polylines as Pipe Runs*
2 From the Alignments menu, choose Set Current Alignment to select the alignment that you associated with the pipe run or that you created from the pipe run.	*Make an Alignment Current*
3 From the Profiles menu, choose Create Profile ➤ Full Profile to create a full profile of the defined alignment.	*Create a Complete Profile*
4 From the Pipes menu, choose Settings ➤ Edit to display the Design Pipes Settings Editor.	*Overview of Changing the Pipe Settings*
5 Click Profile in the Layer Data section to display the Profile Layer Settings dialog box, and review the names to be used for the profile layers.	*Change the Profile Layer Settings for Pipes*
6 From the Pipes menu, choose Conceptual Profile ➤ Import Run to import the run into the profile.	*Import Conceptual Pipe Runs into Profile View*

To draft a conceptual profile pipe run (*continued*)

Steps	*Use* Find *to look up*
7 You can edit the pipes and nodes of the conceptual profile view from the Pipes menu, by choosing Conceptual Profile ➤ Edit Graphical. If you prefer to edit data using a tabular editor, then from the Pipes menu, choose Conceptual Profile ➤ Edit Data.	*Edit Conceptual Pipe Runs in Profile View* *Edit Conceptual Pipe Runs in Profile View Using the Pipe Run Editor*

Editing Pipe Runs Graphically

There are two ways to edit your pipe run in plan and profile views after you lay it out. You can edit it on screen, adjusting the entities that make up the pipe run, or you can edit it in tabular editors. This section describes how you can use the Edit Graphical command to edit a plan view pipe run visually in your drawing.

Key Concepts

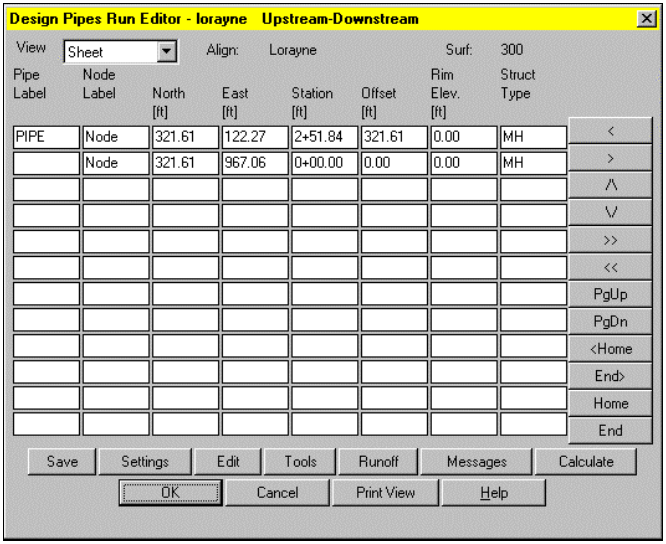
- You can edit the pipe run in plan view. You can add, delete, or move pipe run nodes, and you can edit all the associated database information for each node, including rim and sump elevations.
- You can edit the pipe run in profile view. You can edit nodes or pipes using this method. You can edit the slope of a pipe, starting and ending elevations, and you can edit all the associated database information for the pipe. You can also use the Graph option to graphically edit the pipe run by selecting a point to pass the pipe through.

To edit a conceptual plan pipe run

Steps	*Use* Find *to look up*
1 Define a conceptual plan pipe run.	
2 From the Pipes menu, choose Conceptual Plan ➤ Edit Graphical.	*Edit Conceptual Pipe Runs in Plan View*
3 Select the run that you want to edit by picking it from the screen, or by pressing ENTER and selecting its name from the dialog box. In this example, you will use the DBase option to change a node name.	
4 Move to the node that you want to change by using the Next or Prev options.	
5 Type **DB** to display the Edit Run Node dialog box.	
6 Select the NAME: <name> row.	
7 Type a new name for the node in the Edit box, and then click OK. You can use the DBase option to edit elevations, pipe materials, dimensions, and so on.	
8 Type **S** to save the change to the pipe run database.	

Working with the Pipes Run Editor

You can use the Pipes Run Editor to edit a conceptual pipe run in a dynamic spreadsheet format dialog box. You can use this dialog box to adjust pipe sizing and flow rate parameters of the pipe runs.



You can choose which columns of information that you want to display in the Pipes Run Editor. You can select one of the defined views from the View list to view specific column groupings. For example, you can pick the Node view to view the columns that only pertain to nodes.

Changes that you make in relevant cells of the spreadsheet affect data in other parts of the spreadsheet. For example, increasing the flow rate values in the Pipe Flow column results in increases in the values found in the Pipe Size column, as well as changes to values in the Design Flow, Design Velocity, and Design Depth columns.

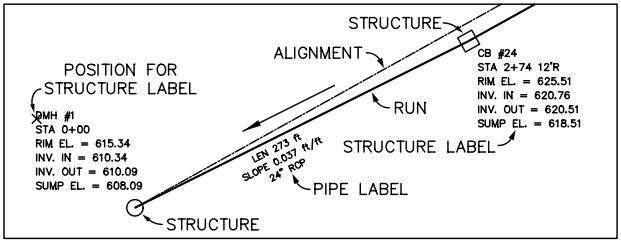
Key Concepts

- Pipe run nodes are listed by northing/easting coordinates, station and offset (if applicable), and node labels.
- Structures at nodes are listed with rim and sump elevations, node and sump drop values, and structure type and dimensions, including structure wall thickness values.

- Pipe segments are listed with pipe size (diameter), start and finish invert elevations, slope, drop, and flow values, as well as roughness coefficients for use in Manning, Darcy-Weisbach, and Hazen-Williams pipe flow calculations formulae. Critical flow and depth values for each pipe segment are listed.
- Contributing flow data from one or two laterals is listed, with lateral names, discharge point invert elevations, and flow values.
- Flow data is listed for each pipe segment, including design flow, design velocity, design depth, and % d/D (percentage full value at a specific design flow rate) values. The wetted and full-flow areas, and wetted and full-flow perimeter values are listed.
- Hydraulic and energy grade line elevations in and out are listed.
- The critical slope, depth, and velocity are listed for each pipe segment, as well as Froude number and flow regime data.
- You can list data from upstream runs, including run name, invert in, and flow values.
- You can list runoff data from an existing surface runoff file.

Drafting Finished Plan Pipe Runs

When you have configured the final details of your pipe run with the Pipes Run Editor, you can draft the finished plan pipe run into your drawing. Illustrative structure blocks and labels for nodes are inserted, and then pipes are drawn and labeled between nodes. The following illustration shows a finish draft plan run detail.



Key Concepts

- You can specify pipe label position, pipe line type, and line text using the Plan Pipe Drafting settings. You can choose which label components to display. You can append prefixes and suffixes to pipe size, slope, material, name, and length labels. You can also specify the precision for size, slope, and length values, and you can add arrows to indicate flow direction.
- You can select node label station, offset, elevation, and name labels in the Plan Node Drafting settings. You can choose to display any of the label components. You can append prefixes and suffixes to node station, right or left offset, and pipe, inverts in and out, sump, and rim elevation labels. You can also specify the precision for station, offset, and pipe, sump, and rim elevation values.
- You can specify the layers for the finished plan pipe run labels.
- You can specify structure label locations by picking points or by entering an offset distance relative to each structure.
- You can rotate structures as they are inserted.
- To properly label pipe runs with the Sheet Manager commands, you must plot the Finished Draft plan view of the pipes (although you do not need to include any textual information such as pipe diameter or invert elevations).

To draft a finished plan pipe run

Steps	*Use* Find *to look up*
1 From the Pipes menu, choose Settings ➤ Edit to display the Pipes Settings Editor dialog box.	
2 Under Pipes Drafting Labels, click Plan to establish the finished plan pipe settings.	*Change the Label Settings for Finished Draft Pipes in Plan View*
3 Under Node Drafting Labels, click Node to establish the finished plan node settings.	*Change the Label Settings for Finished Draft Nodes in Plan View*
4 From the Pipes menu, choose Finish Draft Plan ➤ Draw Pipes, and then select the pipe run. You can select the pipe run from the drawing by clicking on it, or you can press ENTER to display the Defined Runs dialog box where you can select the run.	*Create Finished Draft Runs in Plan View*

To draft a finished plan pipe run (*continued*)

Steps	*Use* Find *to look up*
5 Specify the layers for the finished plan pipe run labels.	
6 Specify the option for placing the structure labels: Picking or Offset. If you choose the picking option, then you will be prompted to locate each structure label as it is drawn.	
7 Specify whether or not you want to rotate each structure as it is inserted in the drawing. The finished plan pipe run is drawn.	

Drafting Finished Profile Pipe Runs

When you have configured the final details of your pipe run with the Pipes Run Editor, you can draft the finished profile pipe run in the current profile. Just as for drafting the finished plan pipe run, illustrative structure blocks and labels for nodes are inserted, and then pipes are drawn and labeled between nodes.

Key Concepts

- You must have a properly defined current profile in the drawing to draft the finished profile pipe run.
- You can specify the pipe label position and slope percentage using the Profile Pipe Drafting settings. You can choose which label components to display. You can append prefixes and suffixes to pipe size, slope, material, name, and length labels. You can also specify the precision for size, slope, and length values, and you can add arrows to indicate flow direction.

- You can specify node label station, offset, elevation, and name labels with the Profile Node Drafting settings. You can choose which label components to display. You can append prefixes and suffixes to node station, right or left offset, and pipe, inverts in and out, sump, and rim elevation labels. You can also specify the precision for station, offset, and pipe, sump, and rim elevation values, as well as the text grouping configuration.
- To properly label pipe runs with the Sheet Manager commands, you must plot the Finished Draft profile view of the pipes (although you do not need to include any textual information such as pipe diameter or invert elevations).

To draft a finished profile pipe run

Steps	*Use* Find *to look up*
1 If you do not have a profile currently drafted in your drawing for the pipe run alignment (or the alignment that you associated with the pipe run), then from the Profile menu, choose Create Profile ➤ Full Profile to draw the profile.	*Create a Complete Profile*
2 From the Pipes menu, choose Settings ➤ Edit to display the Pipes Settings Editor dialog box.	
3 Under Pipes Drafting Labels, click Profile to establish the finished profile pipe settings.	*Change the Label Settings for Finished Draft Pipes in Profile View*
4 Under Node Drafting Labels, click Profile to establish the finished profile node settings.	*Change the Label Settings for Finished Draft Nodes in Profile View*
5 From the Pipes menu, choose Finish Draft Profile ➤ Draw Pipe Run, and then select the pipe run. You can pick the pipe run from the drawing, or you can press ENTER to display the Defined Runs dialog box, where you can select the run. The finished draft profile pipe run is drawn on the existing profile.	*Create Finished Draft Runs in Profile View*

8

Plotting Drawings

You can use the Sheet Manager commands to automate plan, profile, and section sheet plotting.



In this chapter

- Overview of Plotting Drawings
- Working in Model and Paper Space
- Creating Label Styles, Sheet Styles, and Frames
- Setting Up a Plan/Profile Sheet Style
- Creating a Plan/Profile Sheet Series
- Creating a Section Sheet Series

Overview of Plotting Drawings

Sheet Manager commands are used to create paper space sheets for plotting that are based on alignments, profiles, and cross sections model space in your drawing.

Sheet Manager creates three types of plotted sheets:

- **Profile Sheets**: Profile sheets are defined with a single viewport definition. Based on the scale and the size of the viewport, a series of sheets are generated for the alignment's profile.
- **Plan and Profile Sheets**: Plan and Profile sheets are defined with two viewport definitions, one for plan and one for profile. Based on the scale and the size of the viewport, a series of sheets are generated for the alignment and profile. The layout is determined by the length of profile that can be displayed per sheet. The plan view is then aligned to coincide with the profile view.
- **Cross Sections Sheets**: Cross section sheets are defined with a single viewport definition. However, unlike plan and profile sheets, this viewport definition is duplicated many times per sheet based on the number of cross sections that can fit within the sheet style's section frame for the desired scale.

To create plotted sheets for an alignment, you first create a drawing with the plan, plan and profile, or cross section elements in model space. You then select a sheet style to use as a template to generate the sheet series. You can use the sample templates that are included with Sheet Manager as they are, or you can modify them to meet your standards. You can also create new sheet styles from scratch.

Sheet styles are comprised of viewports, frames, and label styles. Label styles, such as profile stationing, are defined in a library of styles and then positioned on the sheets using label frames.

Once you select the sheet style, you generate a series of sheets for the alignment. If it is a plan and profile sheet, rectangles that represent the viewport definitions are plotted in model space so that you can see how the sheets will be laid out before the actual sheets are generated. You can adjust these layouts prior to generating the sheets. You can also modify the profile layouts using the available commands prior to generating sheets.

Once the sheets have been generated, you can use the available commands to fine tune the sheets and you can add additional information. Each sheet is saved to an external file. Finally, you plot the sheets either one at a time, or an entire sheet series using the batch plotting command.

The following is an overview of the key concepts in Sheet Manager:

Sheet Styles

Sheet styles are paper space templates that are used for an alignment. A sheet style is comprised of viewports, frames, and label styles. They are stored in an external folder so that they can be accessed from any drawing. Multiple collections of sheet styles can be created to meet different plot standards.

Viewports: Viewports are AutoCAD Paper Space Viewports with additional properties. These properties include view type (plan, profile, or cross section) and plotted scale.

Frames: Frames are rectangular polylines that are used to position the automatic labeling that occurs when sheets are generated. There are four categories of frames:

- The Label frame is used to position labels to the sides, above or below profiles and cross sections. This is typically information such as station and elevation along the bottom of a profile, or the grid elevations on the sides of the profile.
- View frames are used to position labels directly over the view definition to label information, such as plan view alignment stationing or profile vertical alignment information.
- Table frames are only for cross section sheets and are used to plot area and volume information.
- Section labels are only for cross section sheets and are used to define how cross sections are positioned on the sheet.

Label Styles: Label styles define the different types of annotation plotted on the sheet frames, such as station or elevation labels. The four types of label styles are Text, Block, Distance, and Grid. There are many predefined label styles with Sheet Manager that can be modified, or new styles that can be created.

Sheet Series

There are separate groups of commands for generating series of sheets for plan and profile, profile, or cross sections sheets. A sheet series is defined by a selected sheet style and alignment. The sheet series generates a number of sheet layouts based on the length of profile, or number of sections that can be plotted on an individual sheet to a paper space layout.

Each sheet is saved to an external file. These sheets can be loaded into a paper space layout and plotted. You can make edits to a sheet and then save the sheet back to the external file.

NOTE The term layout is used to define the process of creating a series of sheets in Sheet Manager for a given alignment. It is also used in AutoCAD to define a named paper space layout.

Sheet Tools

Sheet tools are used to make modifications to sheets after they have been generated. They are used to adjust the positioning of viewports for plan or profile, to move entities between model space and paper space, and to update labels based on changes to label styles or the plan, profile, and cross section elements.

Plotting

In addition to the standard plotting of individual sheets, there is a batch plotting command in Sheet Manager to automate the plotting of multiple sheets. For a selected group of sheets in a series, batch plotting loads each sheet into paper space then plots it.

Working in Model Space and Paper Space

When you are working with Sheet Manager commands, you work in both paper space (layout mode) and model space. Model space is where you draw and manipulate your objects. You set up sheets to plot in a layout. For the majority of time you work in a layout when you are setting up sheets to plot the Sheet Manager commands. You switch between model space and layout mode by clicking the Model and Layout tabs at the bottom of the drawing window.

You can do different things depending on which mode you are in. For example, when you are in model space mode, you can create and edit objects in your model space drawing. When you are in layout mode, you can draw entities, such as a sheet border, to be plotted when you plot the drawing.

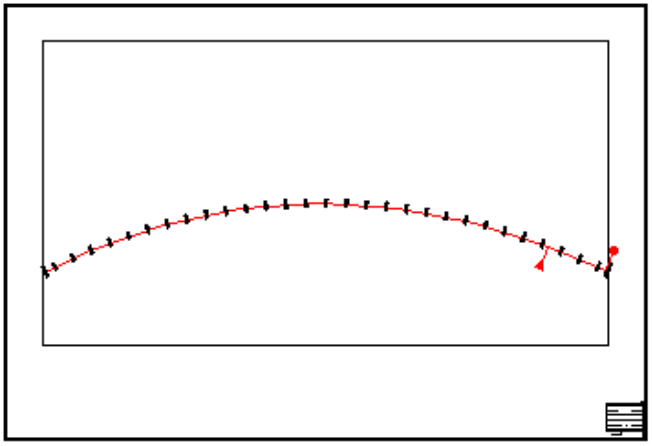
Creating Label Styles, Sheet Styles, and Frames

Customizing your sheet can include changes to the styles used for any annotation, grids, symbols, and so on. A sheet style defines whether a sheet will display a profile view, plan view, or sections from an alignment. After it is defined, the sheet style allows you to automatically generate a sheet with a wide assortment of detailed design annotation. Sheet styles can be saved and reused by everyone working on current and future projects.

Creating Label Styles

To understand how to label and set up sheets with the Sheet Manager commands, you should understand the concept of styles. A style applies specific formatting to the information that is contained in your drawing. For example, you can apply a style to a paragraph in a word processing document to make the paragraph indented or bold.

A label style works in a similar way. A label style contains information about what to label, as well as how. When you set up a label style, you could choose Alignment/Stations as what to label. You could choose design/incremental as how to place these labels. What would show up on the sheet are labels that appear along the alignment at station increments, as shown in the following illustration.



Creating Sheet Styles

A sheet style contains all of the layout and labeling information for the sheets. A sheet style contains a sheet border and title block, viewports, and frames.

Like label styles, sheet styles contain information about what to label, as well as how. They also contain information about how the model space entities will appear on sheets.

When you set up a sheet style, you determine:

- How the parts of your alignment, profile, or sections appear on sheets. For example, you can draw a plan viewport and assign the category Plan to it. This means that this viewport is reserved for plan views of the alignment.
- How the labels appear on the sheets by drawing frames. Frames control the placement of labels on the sheet. If you want labels along an alignment, then draw a frame around the plan viewport.
- What to label by attaching label styles to the frames.
- How to place the labels on the sheet by configuring frame options.

Creating Frames

Frames control where the labels appear on a generated sheet. Frames are part of the sheet style.

There are two parts to using frames. First, you draw frames on the sheet, and then you attach label styles to them.

When you attach label styles to a frame, you define the specific location of the labels. For example, in the illustration of the station labels in the "Creating Label Styles" topic, the label style was attached to the frame around the viewport by using a "Design/Incremental" frame attachment option. This option placed the labels over the design elements (the alignment) at increments based on the stationing.

When you generate the sheet:

- The labels attached to a frame are contained within the frame.
- The labels are positioned within the frame based on label placement options that you set when you attach the labels to the frames.

Setting Up a Plan/Profile Sheet Style

A sheet style is a 1:1 scale paper space sheet template that typically contains a border, a title block, viewports, frames, and label styles. You can customize a sheet style by creating frames and associating label styles with the frames.

Key Concepts

- You work on sheet styles in paper space (layout mode).
- Each plan/profile or plan sheet style has viewports to display plan and profile views of the drawing.
- Section sheets do not use viewports. They use a view frame to display sections.
- Sample sheet styles are included. You can load these sheet styles and customize them, or you can create a sheet style by drawing viewports and frames.

To customize a plan/profile sheet style

Steps	*Use* Find *to look up*
1 Start a new drawing. Give it a unique project name and a unique drawing name.	
2 Load the sheet that you want to edit into paper space from the Sheet Manager menu, by choosing Sheet Styles ➤ Load Sheet Style.	*Load a Sheet Style*
3 From the Sheet Manager menu, choose Sheet Styles ➤ Create Viewport to draw new viewports if necessary.	*Create a Viewport*
4 From the Sheet Manager menu, choose Sheet Styles ➤ Set Viewport Category to set the viewport categories. For example, you can set one viewport so it shows the plan view and one so it shows the profile view. You also set the scale (which should match the drawing that the plan and profile are drafted in).	*Choose a Viewport Category*
5 From the Sheet Manager menu, choose Sheet Styles ➤ Create/Edit Frame to draw frames for annotation.	*Draw a Section/View Frame*

To customize a plan/profile sheet style (*continued*)

Steps	*Use* Find *to look up*
6 From the Sheet Manager menu, choose Sheet Styles ➤ Text Label to edit or to create label styles.	*Create a Text Label*
7 From the Sheet Manager menu, choose Sheet Styles ➤ Create/Edit Frame to attach label styles to the frames.	*Attach Label and Grid Styles to a Frame*
8 From the Sheet Manager menu, choose Sheet Styles ➤ Save Sheet Style to save the sheet style to the sheet style directory.	*Save a Sheet Style*

Creating a Plan/Profile Sheet Series

A sheet series is a group of sheets that is associated with a particular alignment in your drawing. Each sheet shows a specific area of the plan and profile alignment. Creating the series involves laying out the sheets, which determines what part of the plan and profile appear on the sheet, and then generating the sheets.

Key Concepts

- When you lay out the sheet series, rectangles are placed along the current alignment. Each rectangle represents a sheet in the series that is generated. These rectangles are called view definitions.
- You can lay out a sheet series for a plan, plan/profile, or a profile sheet series.
- It is not necessary to use the Layout Sheet Series command when creating a section sheet series.
- Each sheet series has a name and a particular sheet style associated with it. You can choose a predefined sheet style or you can customize a sheet style.
- To correctly label finished draft pipe runs, they must be drafted in the drawing. Textual information does not need to be drafted—only the finish drafted pipes.

To create a plan/profile sheet series		
Steps		***Use*** Find ***to look up***
1	From the Sheet Manager menu, choose Settings to set the Sheet Manager preferences. For plan/profile sheets, you can specify the layer names, whether the sheets are generated with fixed profile stations, and so on.	*Overview of Sheet Manager Settings*
2	In model space, from the Alignments menu, choose Set Current Alignment to select the current alignment.	*Make an Alignment Current*
3	From the Profiles menu, choose Set Current Profile to select the current profile.	*Make a Profile Current*
4	From the Sheet Manager menu, choose Plan/Profile Sheets ➤ Layout Sheet Series to display the Set Current Sheet Series Name dialog box.	*Lay Out a Plan/Profile Sheet Series*
5	Type a name for the new series and then click OK to display the Edit Sheet Series dialog box.	
6	Set up the sheet series options. These options include the sheet style that you want to use to generate the sheet series, the starting sheet number, and the sheet overlap distance.	
7	Click OK to place the view definition rectangles along the alignment. Each view definition represents one sheet that will be created.	
8	Edit the layout, if necessary from the Sheet Manager menu, by choosing Plan/Profile Sheets ➤ Edit Sheet Layout. You can move, rotate and adjust the datum of the view definitions that were placed over the alignment so that each sheet contains the part of the alignment that you want it to.	*Edit a Plan/Profile Sheet Layout*
9	From the Sheet Manager menu, choose Plan/Profile Sheets ➤ Generate Sheet - Series to generate the sheet series.	*Generate a Series of Plan/Profile Sheets*

To create a plan/profile sheet series (*continued*)

Steps	*Use* Find *to look up*
10 You can view one sheet at a time by loading it into paper space. From the Sheet Manager menu, choose Plan/Profile Sheets ➤ Load Sheet - Individual. You can only have one sheet loaded into a layout at a time. If you load another sheet, the first sheet will be removed from the layout. The Sheet Manager ➤ Plan/Profile ➤ Load Sheet – Series command can load up to 255 sheets per drawing. Each sheet is loaded into its own layout.	*Load a Generated Plan/Profile Sheet* *Load a Plan/Profile Sheet Series*
11 From the Sheet Manager menu, choose Plot ➤ Edit Batch Plot Job to select a group of sheets to plot.	*Batch Plot Sheets*
12 From the Sheet Manager menu, choose Plot ➤ Run Batch Plot Job to plot the sheets.	*Run a Batch Plot Job*

Creating a Section Sheet Series

To create a section sheet series, you use frames to display sections on the sheet instead of viewports. Because there are no viewports on a section sheet style, you do not lay out the sheet series like you do with a plan/profile series.

Key Concepts

- A section sheet style must have one Section/View frame and one Section/Section frame. A section sheet style can have any number of label and table frames.
- The easiest way to start generating section sheets is to use a predefined section sheet style. There are predefined sheet styles that you can use in the \data\sheets directory. For example, in \data\sheets\metric there is a cross section sheet named xs100m.dwg that you can use.

The Section Sheet settings are very important when you are generating section sheets. For example, be sure to configure the horizontal scale correctly so that the section swath width that you sampled fits on the sheets.

You can use table frames to position labels on sections sheets that do not have design-specific locations, such as volume calculations.

To create a section sheet series

Steps	*Use* Find *to look up*
1 Create finished ground cross sections using the commands in the Cross Sections menu.	*Overview of Working With Cross Sections*
2 Select the current alignment and profile.	*Make an Alignment Current* *Make a Profile Current*
3 From the Sheet Manager menu, choose Preferences, and then click the Section Preferences button to set the cross section sheet settings. These settings control margins, scales, and volume calculation methods.	*Overview of Cross Section Sheet Preferences*
4 From the Sheet Manager menu, choose Section Sheets ➤ Generate Section Sheets to display the Set Current Series Name dialog box.	*Overview of Sheet Series Generation*
5 Type a new name for the series and then click OK to display the Edit Sheet Series dialog box.	
6 Select the sheet style to use, set the starting sheet number, the starting section number, and the starting and ending stations.	*Generate a Section Sheet Series*
7 Click OK to generate the sheets.	

Index

3D grid, roadway, 87
alignment
 cross sections, 70, 75
 editing, 67
 profile, 63, 64, 67
 superelevating, 83
 transitioning, 79
AutoCAD, running with
 Softdesk programs, 6
breaklines, creating from
 grading object, 23, 25
Civil Design, 2
 documentation, 9
 exiting, 14
 menus, 7
 release 2 features, 8
 running with Land
 Development Desktop, 6
 sample projects, 2
 starting, 6
contours, grading object, 23, 24
cross sections
 creating existing ground, 70
 editing, 77, 78
 finished ground, 75
cul-de-sacs, designing, 59
culverts, designing with calculator, 38
daylighting commands
 creating grading plans, 27
design control
 cross sections, 75, 77
design pipe runs, 90
detention basin storage,
 estimating, 52
detention pond, designing, 29
flowrate, using Manning's n calculator,
 37
footprint, grading object, 17
frames, creating, 109
grading
 developing a grading plan, 16
 finished ground surface, 16
grading object
 breaklines, 23
 calculating volumes, 25
 contours, 23
 creating, 17, 18, 19
 editing, 20, 21, 22
 footprint, 17
 surfaces, 23
grading plans, creating
 using daylighting commands, 27
Grading Wizard, 17, 18
Graphical Peak Discharge Method
 (GPDM), 45
grips, editing grading objects, 21
help, finding, 9
 tutorials, 14
hydrologic analysis in
 site development, 35
hydrologic studies, 34
hydrology calculators, 36
Hydrology Tools, introduction, 34
intersections, designing, 57
label styles, creating, 108
landscape symbols,
 adding to drawing, 31
layout mode, working in, 107
long section. *See* roadway profile
Manning's n gravity
 pipe calculator, 37
menus, loading, 7
model space, working in, 107
online Help
 accessing, 10
 printing entire file, 11

online Help (*continued*)
printing single Help topic, 14
paper space (layout mode), working in, 107
parking lots, adding to drawing, 31
patios and walks, adding to drawing, 31
peak runoff, calculating, 41
pipe runs
conceptual profile, 95
designing, 90
drawing and defining, 91
editing graphically, 96
finished plan, 99
finished profile, 101
importing, 94
Pipes Run Editor, 98
printing
entire Help file, 11
single Help topic, 14
profile
creating existing ground, 63
creating finished ground, 64
editing vertical alignments, 67
Rational Method, 41
reference material, 9
retention pond, designing, 29
roadway
3D grid, 87
cross sections, 70, 75
editing, 67
profile, 63, 64, 67
slope, 81
superelevating, 83
transitioning, 79
runoff
calculating, 41, 45, 48
designing culverts, 38
hydrologic analysis, 35
section sheet series, creating, 113
sections, roadway, 70
Sheet Manager, 104
creating sheet styles, 108
cross section sheets, 104
frames, 109
label styles, 108
plan and profile sheets, 104
Sheet Manager (*continued*)
plotting, 106
profile sheets, 104
sheet tools, 106
sheet series, 106
generating, 111
laying out, 111
plan/profile, 111
section sheet series, 113
sheet styles, 105
creating, 108
definition, 105
frames, 109
setting up plan/profile, 110
site development
hydrologic analysis, 35
slope, roadway, 81
Softdesk programs
creating reports and plans, 2
sports fields, adding to drawing, 31
storm water runoff, estimating, 45
superelevation, roadway, 83
surface data, grading object, 23
surfaces
adding breaklines to, 23
creating from grading object, 23
grading for detention pond, 29
Tabular Hydrograph Method, 45, 48
Technical Release 55. *See* TR-55
Terrain Model Explorer, 35
TR-55
Detention Basin Storage, 52
Graphical Peak Discharge Method, 45
Tabular Hydrograph Method, 45, 48
transition, roadway, 79
tutorial, accessing, 14
vertical alignment. *See* roadway profile
volumes, calculating for grading object, 25, 26
watershed areas
calculating peak runoff, 41
estimating storm water runoff, 45
watershed hydrologic analysis, 35